

✦ This user manual describes all proceedings concerning the operations of this CNC system in detail as much as possible. However, it is impractical to give particular descriptions for all unnecessary or unallowable system operations due to the manual text limit, product specific applications and other causes. Therefore, the proceedings not indicated herein should be considered impractical or unallowable.

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

## Preface

Dear user,

It's our pleasure for your patronage and purchase of this GSK218MC-H CNC system made by GSK CNC Equipment Co., Ltd.

The manual is "Programming and operation", and this manual introduce the method of programming and PLC operation detailedly, please read it seriously before operating.



**Accident may occur by improper connection and operation! This system can only be operated by authorized and qualified personnel. Please carefully read this manual before usage!**

### **Special cautions:**

**The power supply fixed on/in the cabinet is exclusively used for the CNC system made by GSK. It can't be applied to other purposes, or else it may cause serious danger.**

**This manual is reserved by final user.**

All specifications and designs herein are subject to change without further notice.

We are full of heartfelt gratitude to you for supporting us in the use of GSK's products.

## Warning and Precautions

### Warning, note and explanation

This manual contains the precautions to protect user and machine. The precautions are classified as warning and note by safety, and supplementary information is regarded as explanation. Read the warnings, notes and explanations carefully before operation.

### Warning

Personnel may be hurted or equipment be damaged if operations and steps are not observed.

### Note

Equipment may be damaged if operation instructions or steps are not observed by user.

### Explanation

It is used for the supplementary information except for warning and note.

- **Copy right is reserved.**

## Safety Responsibility

### Safety responsibility for manufacture

- Manufacturer should be responsible for eliminated or controlled safety in design and structure of the CNC system and accessories.
- Manufacturer should be responsible for safety of design and structure of the CNC system and accessories.
- Manufacturer should be responsible for information and suggestion for the user.

### Safety responsibility for user

- User should master the safety operation content by learning about and being trained the CNC system safety operation.
- User should be responsible for the danger caused by adding, changing or modifying the CNC system and accessories by itself.
- User should be responsible for the danger because of the mistaken operation, regulation, maintenance, installation and storage..

# CONTENT

|  |           |
|--|-----------|
| <b>CONTENT .....</b>   | <b>V</b>  |
| <b>II PROGRAMMING .....</b>                                    | <b>2</b>  |
| <b>1 General.....</b>  | <b>3</b>  |
| 1.1 Tool movement along workpiece contour —interpolation ..... | 3         |
| 1.2 Feed—Feed function .....                                   | 4         |
| 1.3 Cutting feedrate, spindle speed function .....             | 5         |
| 1.4 Operation instruction—miscellaneous function.....          | 5         |
| 1.5 Tool selection for various machining—Tool function .....   | 5         |
| 1.6 Tool figure and tool motion by program .....               | 6         |
| 1.6.1 Tool length compensation .....                           | 6         |
| 1.6.2 Tool radius compensation .....                           | 6         |
| 1.7 Tool movement range—stroke.....                            | 7         |
| <b>2 Part Program Composition.....</b>                         | <b>9</b>  |
| 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.....                                     | 13        |
| 2.2.2 Subprogram call.....                                     | 13        |
| 2.2.3 Program end .....  | 14        |
| <b>3 Programming Fundamentals.....</b>                         | <b>15</b> |
| 3.1 Controlled axis .....                                      | 15        |
| 3.2 Axis name .....  | 15        |
| 3.3 Coordinate system .....                                    | 15        |
| 3.3.1 Machine coordinate system .....                          | 15        |
| 3.3.2 Reference point.....                                     | 15        |
| 3.3.3 Workpiece coordinate system .....                        | 16        |
| 3.3.4 Absolute programming and relative programming .....      | 17        |
| 3.4 Mode and non-mode .....                                    | 18        |
| 3.5 Decimal point programming .....                            | 19        |
| <b>4 Preparatory Function: G code.....</b>                     | <b>20</b> |
| 4.1 Classification of G code .....                             | 20        |
| 4.2 Simple G code .....  | 23        |
| 4.2.1 Rapid positioning G00.....                               | 23        |
| 4.2.2 Linear interpolation G01.....                            | 24        |
| 4.2.3 Circular (helical) interpolation G02/G03 .....           | 26        |
| 4.2.4 Absolute/ incremental programming G90/G91 .....          | 31        |
| 4.2.5 Dwell(G04).....  | 32        |
| 4.2.6 Unidirectional positioning (G60).....                    | 32        |

**Content**

|        |   |     |
|--------|---|-----|
| 4.2.7  | System parameter online modification (G10).....             | 33  |
| 4.2.8  | Workpiece coordinate system G54~G59.....                    | 34  |
| 4.2.9  | Additional workpiece coordinate system.....                 | 37  |
| 4.2.10 | Machine coordinate system selection G53 .....               | 37  |
| 4.2.11 | Floating coordinate system G92 .....                        | 38  |
| 4.2.12 | Plane selection G17/G18/G19 .....                           | 40  |
| 4.2.13 | Polar coordinate system setup/cancel G16/G15.....           | 40  |
| 4.2.14 | Scaling in plane G51/G50.....                               | 43  |
| 4.2.15 | Coordinate system rotation G68/G69 .....                    | 47  |
| 4.2.16 | Skip function G31 .....                                     | 50  |
| 4.2.17 | Inch/metric conversion G20/G21 .....                        | 52  |
| 4.2.18 | Optional angle chamfering/corner rounding.....              | 52  |
| 4.3    | Reference point G code.....                                 | 54  |
| 4.3.1  | Reference point return G28 .....                            | 54  |
| 4.3.2  | 2nd, 3rd, 4th reference point return G30 .....              | 56  |
| 4.3.3  | Automatic return from reference point G29.....              | 56  |
| 4.3.4  | Reference point return check G27 .....                      | 57  |
| 4.4    | Canned cycle G code .....                                   | 57  |
| 4.4.1  | Rough milling of inner circular groove G22/G23 .....        | 63  |
| 4.4.2  | Fine milling cycle within a circle G24/G25 .....            | 66  |
| 4.4.3  | Outer circle finish milling cycle G26/G32 .....             | 67  |
| 4.4.4  | Rectangular groove rough milling G33/G34.....               | 70  |
| 4.4.5  | Inner rectangular groove finish milling cycle G35/G36 ..... | 73  |
| 4.4.6  | Rectangle outside finish milling cycle G35/G36.....         | 75  |
| 4.4.7  | High-speed peck drilling cycle G73 .....                    | 76  |
| 4.4.8  | Drilling cycle, spot drilling cycle G81 .....               | 78  |
| 4.4.9  | Drilling cycle, counterboring G82.....                      | 80  |
| 4.4.10 | Drilling cycle with chip removal G83 .....                  | 82  |
| 4.4.11 | Right-handed tapping cycle G84 .....                        | 84  |
| 4.4.12 | Left-handed tapping cycle G74.....                          | 86  |
| 4.4.13 | Fine boring cycle G76.....                                  | 88  |
| 4.4.14 | Boring cycle G85 .....                                      | 90  |
| 4.4.15 | Boring cycle G86 .....                                      | 91  |
| 4.4.16 | Boring cycle, back boring cycle G87.....                    | 93  |
| 4.4.17 | Boring cycle G88 .....                                      | 95  |
| 4.4.18 | Boring cycle G89 .....                                      | 97  |
| 4.4.19 | Left-handed rigid tapping G74.....                          | 99  |
| 4.4.20 | Right-handed rigid tapping G84.....                         | 101 |
| 4.4.21 | Peck drilling rigid tapping (chip removal) cycle .....      | 103 |
| 4.4.22 | Canned cycle cancel G80.....                                | 105 |
| 4.5    | Tool compensation G code .....                              | 108 |
| 4.5.1  | Tool length compensation G43, G44, G49 .....                | 108 |
| 4.5.2  | Tool radius compensation G40/G41/G42 .....                  | 111 |
| 4.5.3  | Explanation of tool radius compensation .....               | 118 |
| 4.5.4  | Corner offset circular interpolation (G39) .....            | 134 |
| 4.5.5  | Tool offset value and number input by program (G10) .....   | 136 |

|          |  |            |
|----------|--|------------|
| 4.6      | Feed G code .....  | 137        |
| 4.6.1    | Feed mode G64/G61/G63 .....                                | 137        |
| 4.6.2    | Automatic override for inner corners (G62) .....           | 138        |
| 4.7      | Macro G code .....   | 140        |
| 4.7.1    | Custom macro.....  | 140        |
| 4.7.2    | Macro variables.....                                       | 140        |
| 4.7.3    | Custom macro call .....                                    | 147        |
| 4.7.4    | Operation and transfer instruction.....                    | 147        |
| 4.7.5    | Examples for custom macro.....                             | 157        |
| <b>5</b> | <b>Miscellaneous Function M code .....</b>                 | <b>159</b> |
| 5.1      | M codes controlled by PLC .....                            | 160        |
| 5.1.1    | Forward and reverse rotation instructions (M03, M04) ..... | 160        |
| 5.1.2    | Spindle stop (M05).....                                    | 161        |
| 5.1.3    | Cooling on and off (M08, M09) .....                        | 161        |
| 5.1.4    | A axis release and clamping (M10, M11) .....               | 161        |
| 5.1.5    | Tool release and clamping (M16, M17) .....                 | 161        |
| 5.1.6    | Spindle orientation (M19) .....                            | 161        |
| 5.1.7    | Tool search instruction (M21, M22) .....                   | 161        |
| 5.1.8    | Magazine rotation instruction (M23, M24) .....             | 161        |
| 5.1.9    | Rigid tapping (M29) .....                                  | 161        |
| 5.1.10   | Lubricating on and off (M32, M33) .....                    | 162        |
| 5.1.11   | Helical chip remover on and off (M35, M36) .....           | 162        |
| 5.1.12   | Chip remove hydrovalve ON/OFF (M26, M27) .....             | 162        |
| 5.1.13   | Spindle blowing on and off (M44, M45) .....                | 162        |
| 5.1.14   | Auto tool change start and end (M50, M51) .....            | 162        |
| 5.1.15   | Tool judging after tool change (M53) .....                 | 162        |
| 5.1.16   | Check if spindle has a tool (M55).....                     | 162        |
| 5.2      | M codes used by program.....                               | 162        |
| 5.2.1    | Program end and return (M30, M02) .....                    | 163        |
| 5.2.2    | Program dwell (M00) .....                                  | 163        |
| 5.2.3    | Program optional stop (M01) .....                          | 163        |
| 5.2.4    | Subprogram calling (M98) .....                             | 163        |
| 5.2.5    | Program end and return (M99) .....                         | 163        |
| <b>6</b> | <b>S codes for Spindle Function.....</b>                   | <b>164</b> |
| 6.1      | Spindle analog control.....                                | 164        |
| 6.2      | Spindle switch volume control.....                         | 164        |
| 6.3      | Constant surface speed control (G96/G97).....              | 165        |
| <b>7</b> | <b>Feed Functions F code .....</b>                         | <b>168</b> |
| 7.1      | Traverse .....   | 168        |
| 7.2      | Cutting feedrate .....                                     | 168        |
| 7.2.1    | Feed per minute (G94) .....                                | 169        |
| 7.2.2    | Feed per revolution (G95) .....                            | 169        |
| 7.3      | Tangential speed control .....                             | 170        |
| 7.4      | Feedrate override keys .....                               | 170        |
| 7.5      | Auto acceleration/deceleration.....                        | 171        |

|            |   |            |
|------------|---|------------|
| 7.6        | Acceleration/deceleration for corner of a block .....   | 172        |
| <b>8</b>   | <b>Tool Function .....</b>  | <b>173</b> |
| 8.1        | Tool function .....   | 173        |
| <b>III</b> | <b>OPERATION.....</b>   | <b>174</b> |
| <b>1</b>   | <b>Operator Panel .....</b>   | <b>175</b> |
| 1.1        | Panel layout.....   | 175        |
| 1.2        | Explanation of the panel function.....  | 175        |
| 1.2.1      | LCD area .....  | 175        |
| 1.2.2      | Edit area .....   | 176        |
| 1.2.3      | Screen operation keys.....  | 176        |
| 1.2.4      | Control area.....   | 178        |
| <b>2</b>   | <b>System Power On/Off and Safety Operations .....</b>  | <b>182</b> |
| 2.1        | System power on.....  | 182        |
| 2.2        | System power off.....   | 182        |
| 2.3        | Safety operations.....  | 183        |
| 2.3.1      | Reset operation .....   | 183        |
| 2.3.2      | Emergency stop.....   | 183        |
| 2.3.3      | Feed hold.....  | 184        |
| 2.4        | Cycle start and feed hold .....   | 184        |
| 2.5        | Overtravel protection .....   | 184        |
| 2.5.1      | Hardware overtravel protection.....   | 184        |
| 2.5.2      | Software overtravel protection .....  | 185        |
| 2.5.3      | Release of the overtravel alarm.....  | 185        |
| 2.6        | Storage stroke detection .....  | 185        |
| <b>3</b>   | <b>Interface Display as well as Data Modification and Setting .....</b>                         | <b>189</b> |
| 3.1        | Position display .....  | 189        |
| 3.1.1      | Four types of position display .....  | 189        |
| 3.1.2      | The display of the run time, part count, programming speed and override, actual speed etc. .... | 191        |
| 3.1.3      | Relative coordinate clearing and mediating .....  | 193        |
| 3.2        | Program display.....  | 194        |
| 3.3        | System display.....   | 198        |
| 3.3.1      | Parameter display, modify and setting.....  | 198        |
| 3.3.2      | Modification and setting of the parameter values .....  | 200        |
| 3.3.3      | Macro variable display, modification and setting .....  | 201        |
| 3.3.2      | Modification and setting of the offset value.....   | 202        |
| 3.4        | Setting display .....   | 203        |
| 3.4.1      | Setting page .....  | 203        |
| 3.4.2      | Coordinate setting interface.....   | 204        |
| 3.4.3      | Backup, restore and transfer of the data.....   | 206        |
| 3.4.4      | Password authority setting and modification.....  | 207        |
| 3.5        | Graphic display .....   | 208        |
| 3.6        | Diagnosis display.....  | 210        |
| 3.6.1      | Diagnosis data display.....   | 210        |

|          |   |            |
|----------|---|------------|
| 3.6.2    | Signal viewing .....  | 212        |
| 3.7      | Alarm display .....   | 213        |
| 3.8      | PLC display .....   | 215        |
| 3.9      | Index display .....   | 217        |
| <b>4</b> | <b>Manual Operation .....</b>                                   | <b>223</b> |
| 4.1      | Coordinate axis movement .....                                  | 223        |
| 4.1.1    | Manual feed .....   | 223        |
| 4.1.2    | Manual rapid traverse .....                                     | 223        |
| 4.1.3    | MANUAL feedrate and manual rapid traverse speed selection ..... | 223        |
| 4.1.4    | Manual intervention.....  | 224        |
| 4.2      | Spindle control .....   | 225        |
| 4.2.1    | Spindle CCW .....   | 225        |
| 4.2.2    | Spindle CW .....  | 226        |
| 4.2.3    | Spindle stop .....  | 226        |
| 4.2.4    | Spindle auto gear shift .....                                   | 226        |
| 4.3      | Other manual operations.....                                    | 227        |
| 4.3.1    | Cooling control .....   | 227        |
| 4.3.2    | Lubricating control.....  | 227        |
| 4.3.3    | Chip removal.....   | 227        |
| 4.3.4    | Work light control .....  | 227        |
| <b>5</b> | <b>Step Operation .....</b>                                     | <b>228</b> |
| 5.1      | Step feed.....  | 228        |
| 5.1.1    | Selection of moving amount.....                                 | 228        |
| 5.1.2    | Selection of moving axis and direction .....                    | 228        |
| 5.1.3    | Step feed explanation .....                                     | 229        |
| 5.2      | Step interruption .....   | 229        |
| 5.3      | Auxiliary control in Step mode.....                             | 229        |
| <b>6</b> | <b>MPG Operation .....</b>                                      | <b>230</b> |
| 6.1      | MPG feed.....   | 230        |
| 6.1.1    | Moving amount selection .....                                   | 230        |
| 6.1.2    | Selection of moving axis and direction .....                    | 230        |
| 6.1.3    | Explanation of MPG feed .....                                   | 231        |
| 6.2      | Control in MPG interruption.....                                | 232        |
| 6.2.1    | MPG interruption operation .....                                | 232        |
| 6.2.2    | Relation of MPG interruption with other functions .....         | 233        |
| 6.3      | Auxiliary control in MPG mode.....                              | 233        |
| <b>7</b> | <b>Auto Operation .....</b>                                     | <b>234</b> |
| 7.1      | Selection of the auto run programs .....                        | 234        |
| 7.2      | Auto run start.....   | 234        |
| 7.3      | Auto run stop.....  | 235        |
| 7.4      | Auto running from an arbitrary block .....                      | 236        |
| 7.5      | Dry run .....   | 236        |
| 7.6      | Single block running.....                                       | 237        |
| 7.7      | Running with machine lock .....                                 | 237        |
| 7.8      | Running with M.S.T. lock.....                                   | 238        |

**Content**

|                   |  |            |
|-------------------|--|------------|
| 7.9               | Feedrate and rapid override in auto run.....             | 238        |
| 7.10              | Spindle override in auto run.....                        | 239        |
| 7.11              | Background edit in auto run.....                         | 239        |
| <b>8</b>          | <b>MDI Operation.....</b>                                | <b>241</b> |
| 8.1               | MDI instructions input.....                              | 241        |
| 8.2               | Run and stop of MDI instructions.....                    | 242        |
| 8.3               | Words modification and clearing of MDI instructions..... | 242        |
| 8.4               | Modes changing.....                                      | 242        |
| <b>9</b>          | <b>Machine Zero Operation.....</b>                       | <b>243</b> |
| 9.1               | Conception of machine zero.....                          | 243        |
| 9.2               | Steps for machine zero.....                              | 244        |
| 9.3               | Machine zero steps by program.....                       | 244        |
| <b>10</b>         | <b>Edit Operation.....</b>                               | <b>246</b> |
| 10.1              | Program edit.....  | 246        |
| 10.1.1            | Program creation.....                                    | 247        |
| 10.1.2            | Deletion of a single program.....                        | 252        |
| 10.1.3            | Deletion of all programs.....                            | 252        |
| 10.1.4            | Copy of a program.....                                   | 253        |
| 10.1.5            | Copy and paste of blocks.....                            | 253        |
| 10.1.6            | Cut and paste of block.....                              | 254        |
| 10.1.7            | Replacement of the blocks.....                           | 254        |
| 10.1.8            | Rename of a program.....                                 | 254        |
| 10.1.9            | Program restart.....                                     | 255        |
| 10.2              | Program management.....                                  | 256        |
| 10.2.1            | Program directory search.....                            | 256        |
| 10.2.2            | Number of the program stored.....                        | 256        |
| 10.2.3            | Memory capacity.....                                     | 257        |
| 10.2.4            | Viewing of the program list.....                         | 257        |
| 10.2.5            | Program lock.....  | 257        |
| <b>11</b>         | <b>Communication.....</b>                                | <b>258</b> |
| 11.1              | Serial communication.....                                | 258        |
| 11.1.1            | Program start.....                                       | 258        |
| 11.1.2            | Function introduction.....                               | 258        |
| 11.1.3            | Software usage.....                                      | 259        |
| 11.2              | USB communication.....                                   | 263        |
| 11.2.1            | General and precautions.....                             | 263        |
| 11.2.2            | USB flash disk entry.....                                | 263        |
| 11.2.3            | USB part program operation steps.....                    | 263        |
| 11.2.4            | DNC processing operation steps.....                      | 264        |
| 11.2.5            | USB flash disk system exit.....                          | 264        |
| 11.2.6            | Remarks for USB flash disk model.....                    | 264        |
| <b>APPENDIX 1</b> | <b>.....</b>   | <b>266</b> |
| 1                 | Bit parameter.....                                       | 267        |
| 2                 | Number parameter.....                                    | 288        |

|                         |            |
|-------------------------|------------|
| <b>APPENDIX 2 .....</b> | <b>317</b> |
| <b>Alarm List .....</b> | <b>317</b> |



## 1. Overview

This manual is comprised by following parts:

### **I Programming**

It describes program structure of GSK218MC-H CNC system and elementary knowledge of programming, G functions and the programming format, characteristics and restrictions by NC language.

### **II Operation**

It describes **GSK218MC-H** CNC system each window and setting, 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 GSK218MC-H CNC system.**

# II PROGRAMMING

# 1 General

## 1.1 Tool movement along workpiece contour —interpolation

### 1) Tool movement along a straight line

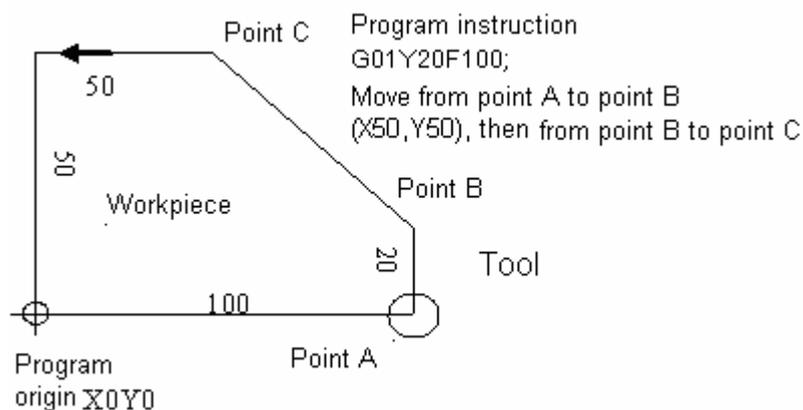


Fig. 1-1-1

### 2) Tool movement along an arc

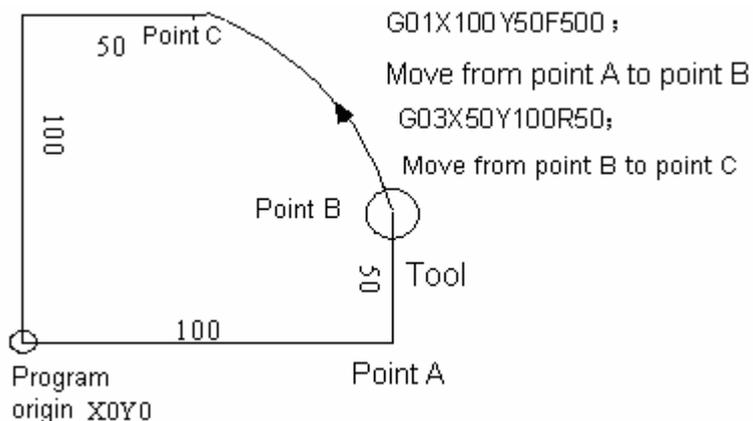
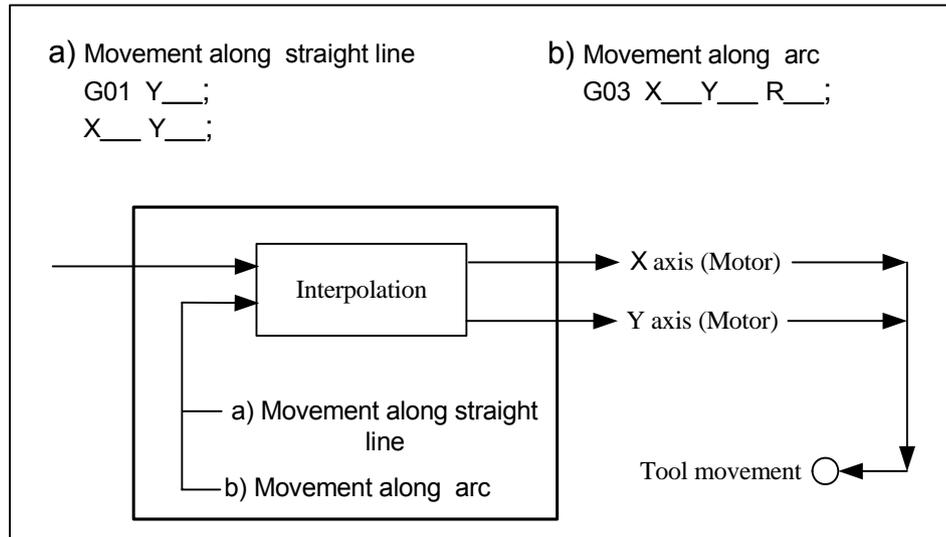


Fig. 1-1-2

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.

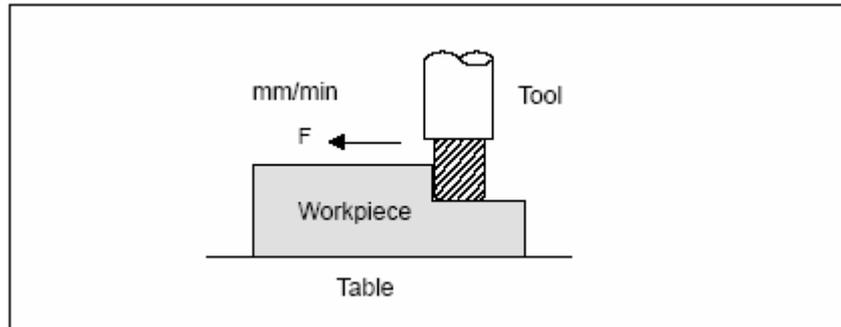


**Fig.1-1-3**

**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

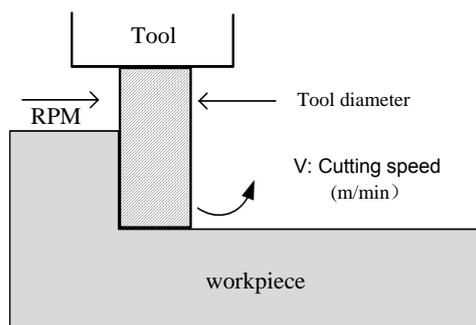
The feedrate specification is called feed function.



**Fig. 1-2-1**

To specify a speed to machine the part by tool is called feed and the machine speed is instructed by a numerical value. For example, the program instruction is F150 if tool feeds by 150mm/min.

### 1.3 Cutting feedrate, spindle speed function



**Fig. 1-3-1**

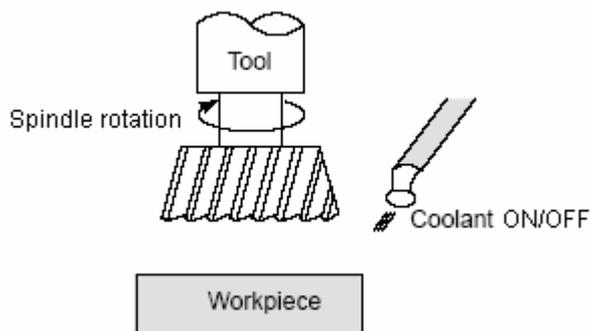
The speed of tool relative to workpiece in cutting is called cutting feedrate. It can be instructed by spindle speed ---unit MM/Rev) by CNC.

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**

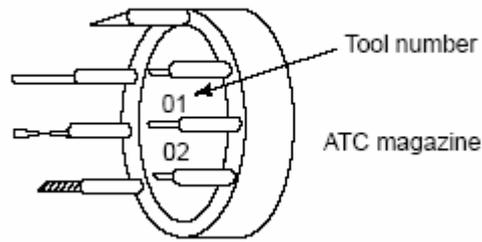
The programs or machine switch actions controlled by system NC instructions are called miscellaneous functions, which are instructed by M code.

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.



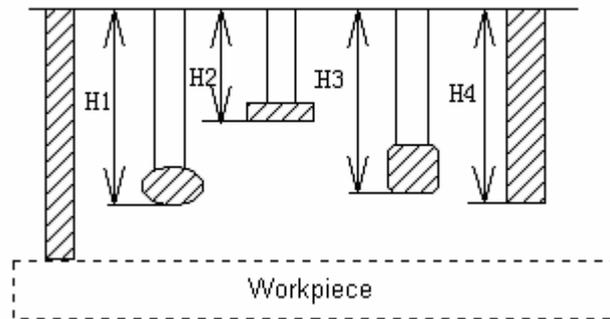
**Fig. 1-5-1**

Example When the tool is placed at location 01 in the ATC magazine, the tool can be selected by specifying T01 code. 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 G0Z0 are executed in a same coordinate system, because tools have different tool lengths, the distances from tool end to workpiece are different. So 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 machining path 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 run 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.

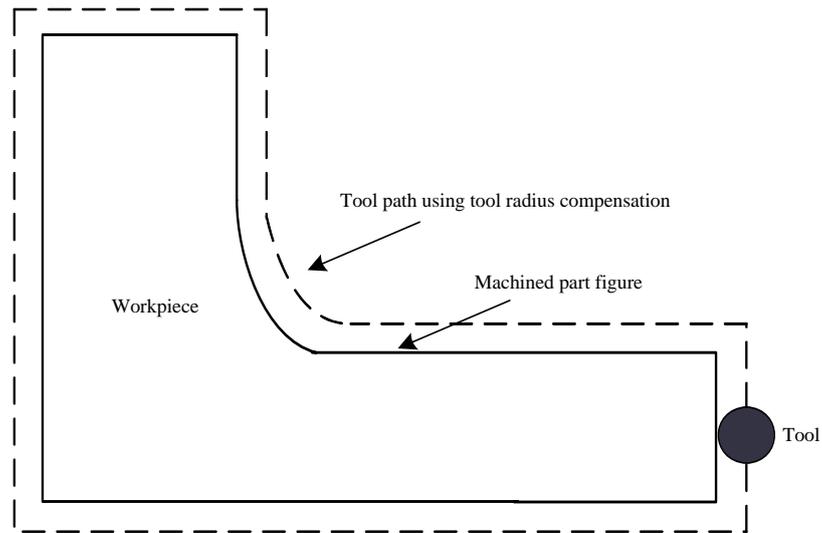


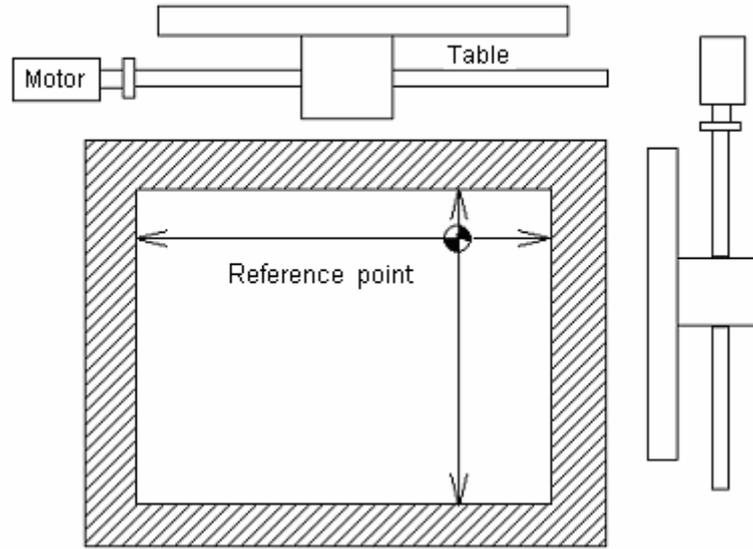
Fig.1-6-2-1

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 travel 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.

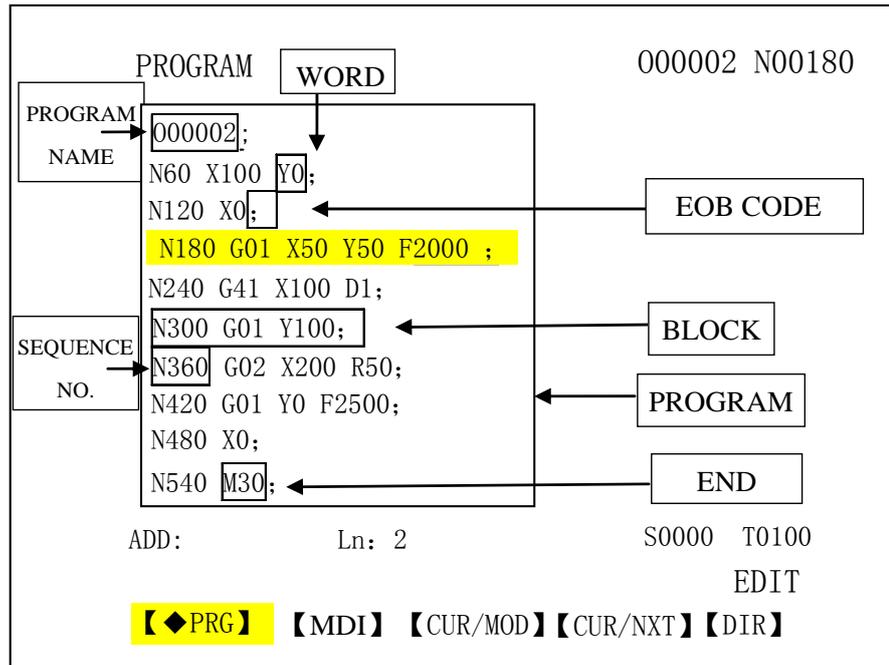


Fig. 2-1-1 Program structure

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.

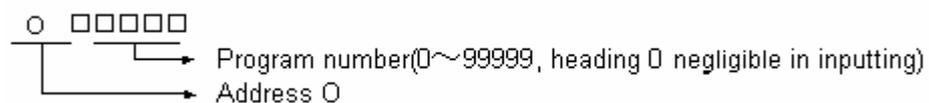


Fig. 2-1-1-1 Program name composition

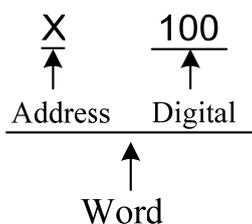
### 2.1.2 Sequence number and block

A program is consisted by 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 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 **N0. 32#6**.

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

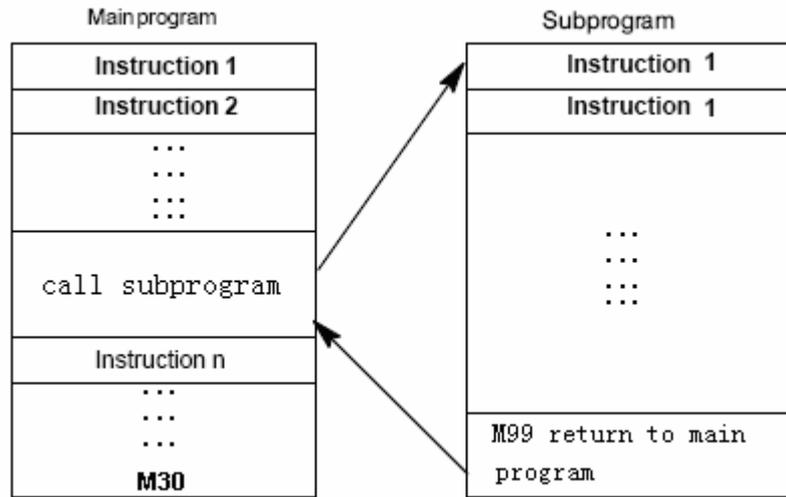
| Address | Range                     | Meaning                                       |
|---------|---------------------------|---|
| O       | 0~99999                   | Program name                                  |
| N       | 0~99999                   | Sequence number                               |
| 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) | Arc center vector in X axis relative to start |

| Address | Range                         | Meaning  |
|---------|-------------------------------|--|
|         |                               | point  |
| J       | -99999.999~99999.999 (mm)     | Arc center vector in Y axis relative to start point                          |
| K       | -99999.999~99999.999 (mm)     | Arc center vector in Z axis relative to start point                          |
| 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  |
| 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.

## 2.2 General structure of a program

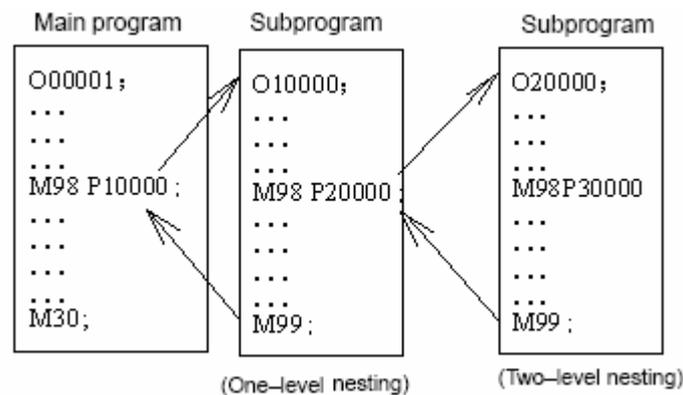
The program is classified for main program and subprogram. Generally, the CNC system is acutated by the main program. If the main program contains the subprogram call, the CNC system acts by the subprogram. If the subprogram contains the instruction of returning to main program, the CNC system returns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



**Fig.2-2-1 Program execution sequence**

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、 No. 90000~99999 subprograms are the system reserved programs, if they are called, they can be executed, and can not 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

| Item                             | GSK218MC-H       |
|----------------------------------|------------------|
| 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 number parameter No.5. The additional axis names are set by number parameter No.6 accordingly, such as A, B, C.

### 3.3 Coordinate system

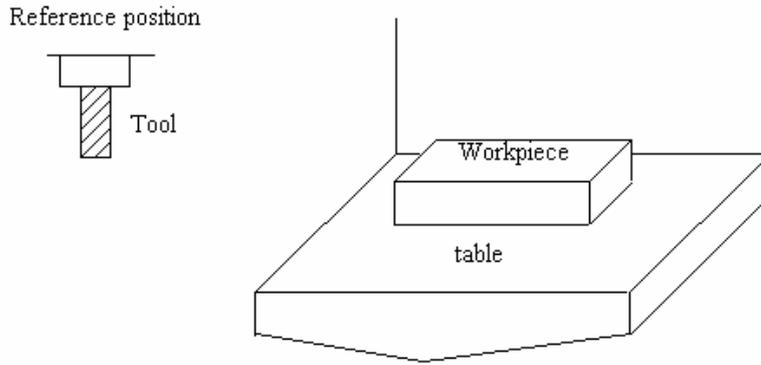
#### 3.3.1 Machine coordinate system

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 is on. Once set, it remains unchanged till the power off, system reset or emergency stop.

This system uses right-hand Cartesian coordinate system. The vertical 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.



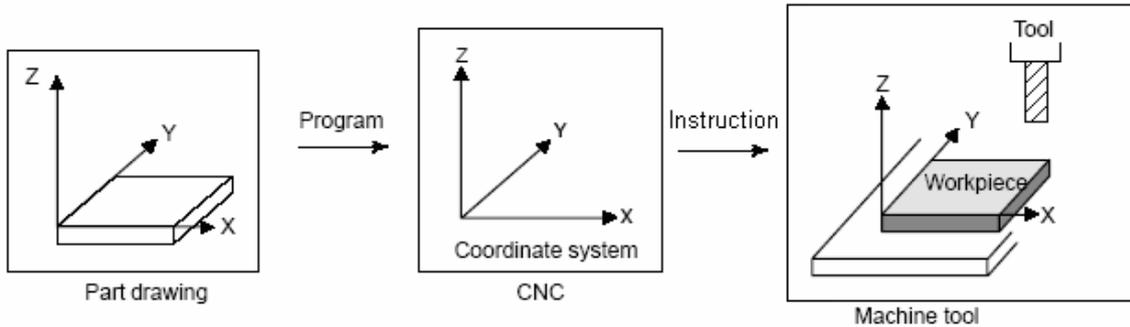
**Fig. 3-3-2-1**

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

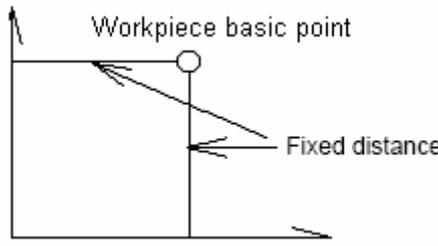
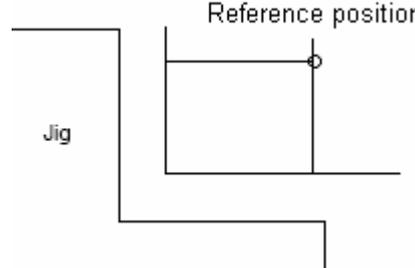
The coordinate system used for workpiece machining is called workpiece coordinate system (or part coordinate system), which is preset by CNC system (to set workpiece coordinate system).



**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 part 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 instructions 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> |

Workpiece coordinate system should be set for 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:

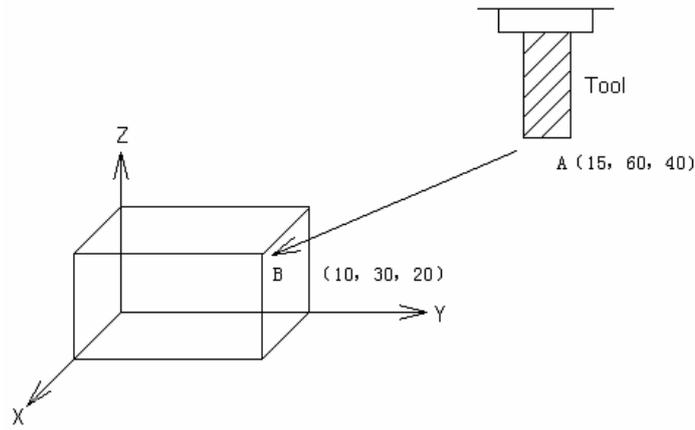
1. By G92, see 4.2.11 for details.
2. By G code from 54 to 59, see 4.2.8 for details.

### 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.



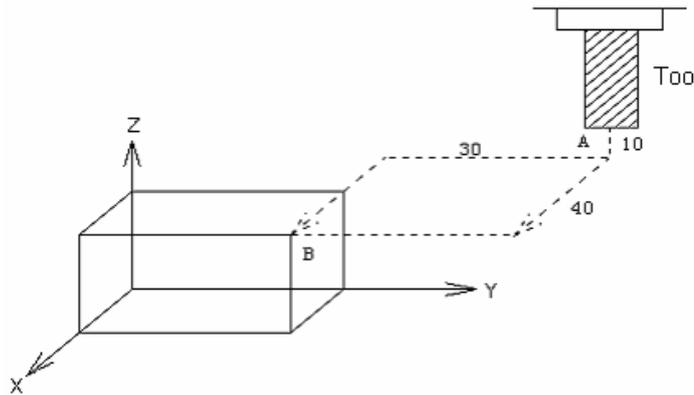
**Fig.3-3-4-1**

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. 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 X20 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.

➤ 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-mode 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 min. moving unit 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-mode G code | Effective in the block in which it is specified              |
| mode G code     | Effective till another G code of the same group is specified |

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

```
G01 X __ ;
Z ____ ; G01 effective
X ____ ; G01 effective
G00 Z____; G00 effective
```

**Note:** Refer to system parameter 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                |       | G10 L_N_P_R_   | Programmable data input  |
| *G11               |       | 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<br>G18<br>G19 | 02    | Write in with other program in block,<br>used for circular interpolation and tool<br>radius compensation | XY plane selection<br>ZX plane selection<br>YZ plane selection |
| G20                | 06    | Specified by a single block at the   | Inch input   |

|      |       |  |  |   |                                     |
|------|-------|--|--|---|-------------------------------------|
| *G21 |       | program  |  | 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  |  | CW inner circular groove rough milling            |                                     |
| G24  |       | G24_X_Y_Z_R_I_J_D_F_K  |  | CCW fine milling cycle within a circle            |                                     |
| G25  |       | G25_X_Y_Z_R_I_J_D_F_K  |  | CW fine milling cycle within a circle             |                                     |
| G26  |       | G26_X_Y_Z_R_I_J_D_F_K  |  | CCW outer circle fine milling cycle               |                                     |
| G27  |       | 00   | G27  | X_Y_Z_  | Reference point return detection    |
| G28  | G28   |  | Reference point return   |   |                                     |
| G29  | G29   |  | Return from reference point  |   |                                     |
| G30  | G30Pn |  | 2 <sup>nd</sup> , 3 <sup>rd</sup> , 4 <sup>th</sup> reference point return |   |                                     |
| G31  | G31   |  | Skip function  |   |                                     |
| G32  | 09    | G32_X_Y_Z_R_I_J_D_F_K  |  | CW outer circle finish milling cycle              |                                     |
| G33  |       | G33_X_Y_Z_R_I_L_W_Q_V_U_D_F_K  |  | CCW rectangular groove rough milling              |                                     |
| G34  |       | G34_X_Y_Z_R_I_L_W_Q_V_U_D_F_K  |  | CW rectangular groove rough milling               |                                     |
| G35  |       | G35_X_Y_Z_R_I_J_L_U_D_F_K  |  | CCW inner rectangular groove finish milling cycle |                                     |
| G36  |       | G36_X_Y_Z_R_I_J_L_U_D_F_K  |  | CW inner rectangular groove finish milling cycle  |                                     |
| G37  |       | G37_X_Y_Z_R_I_J_L_U_D_F_K  |  | CCW rectangle outside finish milling cycle        |                                     |
| G38  |       | G38_X_Y_Z_R_I_J_L_U_D_F_K  |  | CW rectangle outside finish milling cycle         |                                     |
| G39  | 00    | G39  | I_J_; I_J_; J_K_ or G39  | Corner offset circular interpolation              |                                     |
| *G40 | 07    | G17  | G40  | D_X_Y_  | Tool radius compensation cancel     |
| G41  |       | G18  | G41  | D_X_Z_  | Left-hand tool radius compensation  |
| G42  |       | G19  | G42  | D_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                           |                                     |

**Programming**

|      |                         |   |   |
|------|-------------------------|---|---|
| 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               |
| 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 (part of G code are determined by bit parameter NO: 31#0~7) .**

**3、 G codes except G10, G11, G92 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 mode, 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

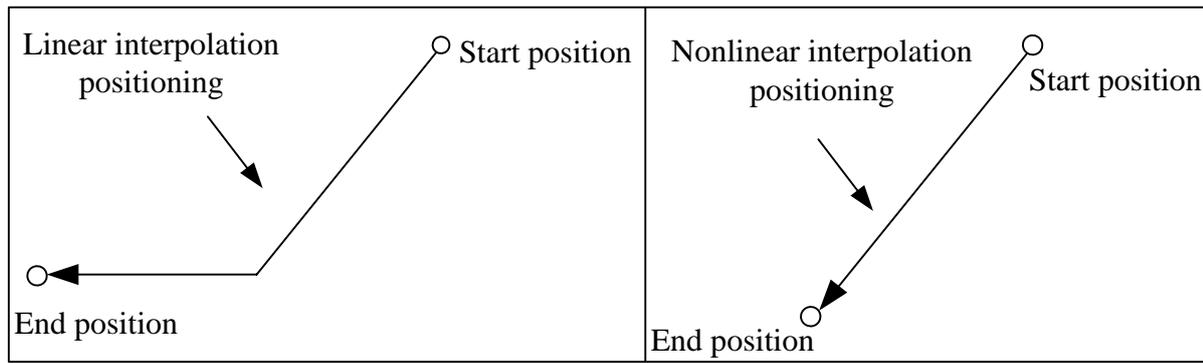
### 4.2.1 Rapid positioning G00

**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. It is set by bit parameter **NO:12#1** and uses the following two paths.( Fig. 4-2-1-1).

1. Linear interpolation positioning: The tool path is the same as linear interpolation (G01) . The tool is positioned within the shortest 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.

**Programming**



**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 are identical with G0.
- 4 G0 speed for X, Y, Z , 4<sup>th</sup> axis is set by number parameter P88~P91.

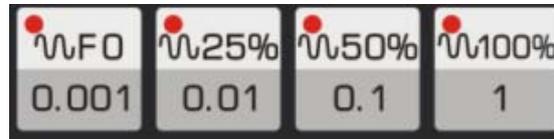
**Restrictions**

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 tool collision.

**4.2.2 Linear interpolation G01**

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

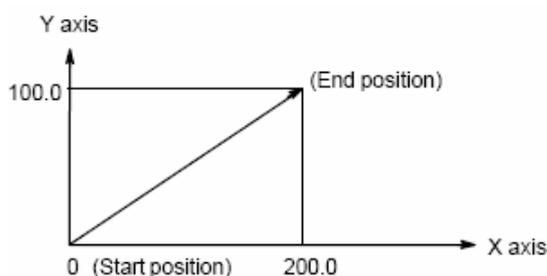
**Function:** The tool moves along a line to the specified position at the feedrate (mm/min)

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)



```
G01 X200 Y100 F200 ;
```

**Note:**

Each axis feedrate is as following:  
 G01 X $\alpha$  Y $\beta$  Z $\gamma$ Ff ;  
 In this block:

Feedrate in X axis:  $F_x = \frac{\alpha^2}{L} \times f$

Feedrate in Y axis:  $F_y = \frac{\beta^2}{L} \times f$

Feedrate in Z axis:  $F_z = \frac{\gamma^2}{L} \times f$

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

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 and 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

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

Arc in ZX plane

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

Arc in YZ plane

```
G02      R_
G19     Y_Z_  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 | G02         | CW                            |

|   | direction                                     | G03                        | CCW  |
|---|---|----------------------------|--|
| 3 | G90<br>End position<br>G91                    | Two axes of X,Y, Z<br>axis | End point coordinate in<br>workpiece coordinate system |
|   |   | Two axes of X,Y, Z<br>axis | Coordinate of end point<br>relative to start point     |
| 4 | Distance from start point<br>to circle center | Two axes of I,J, K<br>axis | Coordinate of circle center<br>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.

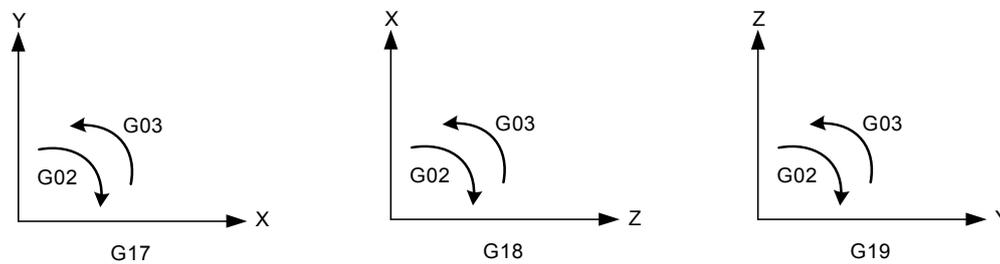


Fig. 4-2-3-1

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.

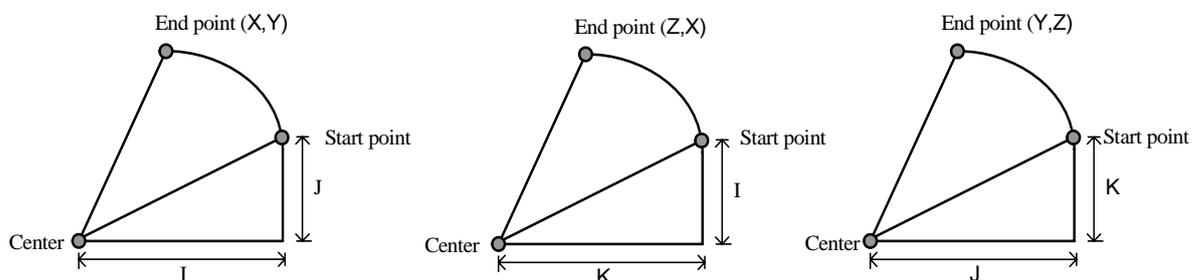


Fig. 4-2-3-2

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\_ ;

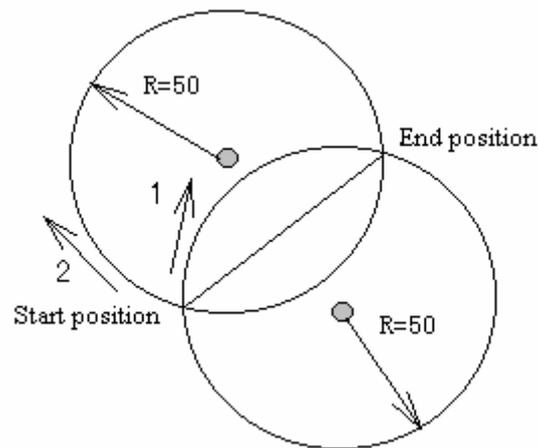
- Two arcs can be drawn out as following, one arc is more than 180°, the other one is less than 180°. The radius of the arc more than 180° should be specified by a negative value.

(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**

- The arc equal to 180° 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°, the positive or negative value of R doesn't affect the arc path.**

- The arc equal to 360° can only be programmed by I, J, K.

(Program example)

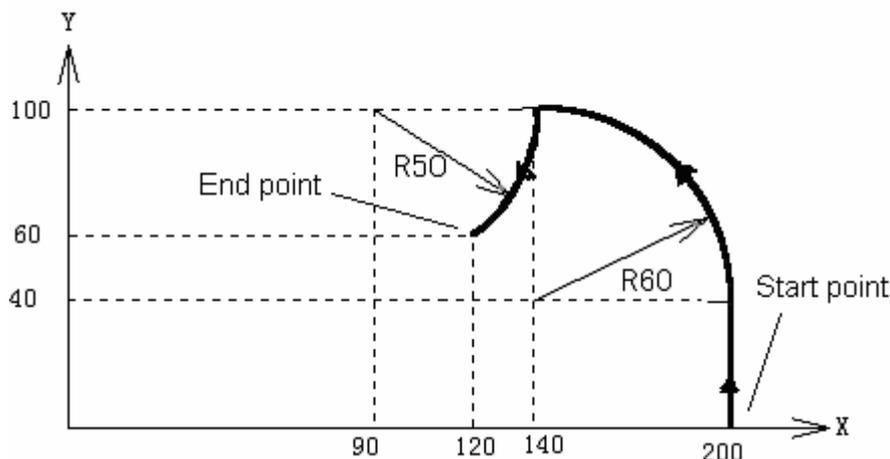


Fig. 4-2-3-4

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**

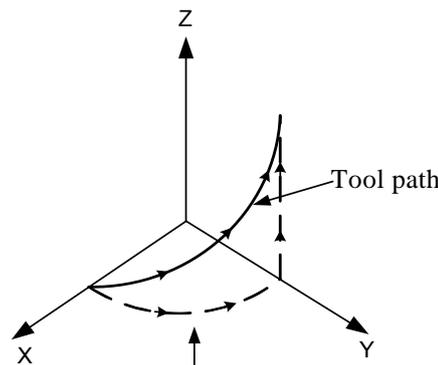
**Format:** G02/G03

|                 |   |                     |   |       |  |
|-----------------|---|---------------------|---|-------|--|
| Arc of XY plane |   |                     |   |       |  |
| G17             | $\left\{ \begin{matrix} G02 \\ G03 \end{matrix} \right\}$ | $X_p - Y_p - Z_p -$ | $\left\{ \begin{matrix} I - J - \\ R - \end{matrix} \right\}$ | $F -$ |  |
| Arc of ZX plane |   |                     |   |       |  |
| G18             | $\left\{ \begin{matrix} G02 \\ G03 \end{matrix} \right\}$ | $X_p - Y_p - Z_p -$ | $\left\{ \begin{matrix} I - K - \\ R - \end{matrix} \right\}$ | $F -$ |  |
| Arc of YZ plane |   |                     |   |       |  |
| G19             | $\left\{ \begin{matrix} G02 \\ G03 \end{matrix} \right\}$ | $X_p - Y_p - Z_p -$ | $\left\{ \begin{matrix} J - K - \\ R - \end{matrix} \right\}$ | $F -$ |  |

**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:**



The feedrate along the circumference of two circular interpolated axes is the specified feedrate

**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 two circular interpolation axes are specified

A moving axis that is not circular interpolation axis is added as for the instruction method, and F instruction specifies the feedrate along an arc. So the feedrate of this linear axis is as following:

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

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.

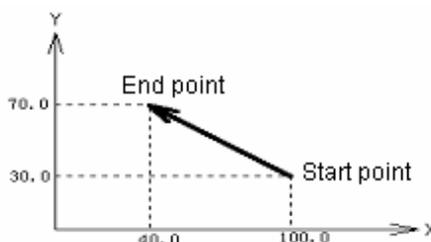


Fig. 4-2-4-1

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.

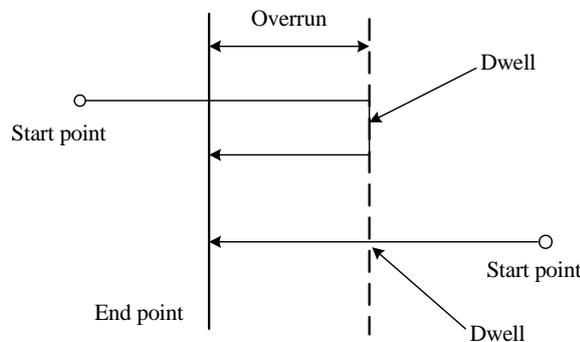
|            |          |              |                   |
|------------|----------|--------------|-------------------|
| <b>G04</b> | <b>X</b> | 0~9999.999   | X for second      |
|            | <b>P</b> | 0~99999.9999 | P for millisecond |

**Explanation:**

- 1 G04 is non-modal instruction, which is only effective in current line.
- 2 If parameters 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\_



**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, 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

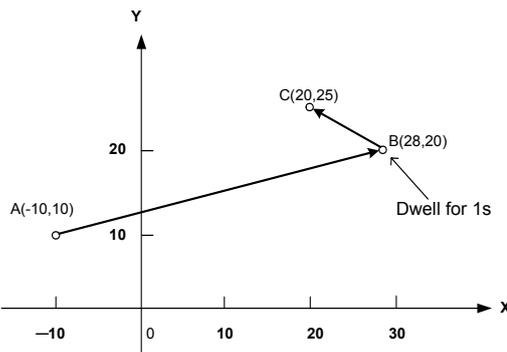


Fig. 4-2-6-2

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: symbol of parameter P335~P338 signifies unidirectional positioning direction; the value signifies overtravel value.

Note 2: overtravel value >0, positioning direction is positive(+).

Note 3: overtravel value <0, positioning direction is negative(-).

Note 4: overtravel value <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 tool radius, length offset, external zero offset,

## Programming

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 for 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 is effective by only restarting after power-off can not be modified by G10.
- 6、 G20、 G21 can not be modified by G10 online.
- 7、 When G10 modifies external zero offset, workpiece zero offset, additional workpiece zero offset or tool offset online, if in G91 module, offset instructed by system and current offset wil be repeated
- 8、 G10 module would be cancelled when M00, M01, M02, M30, M99, M98, M06 is being executed.

### 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. In starting-up, the system displays workpiece coordinate system G54~G59 has been executed before power off or displays additional workpiece 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 alter.

e.g. 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, 30        | 130, 130, 60       |
| X120 Z80           | 120, 100, 80        | 150, 130, 110      |

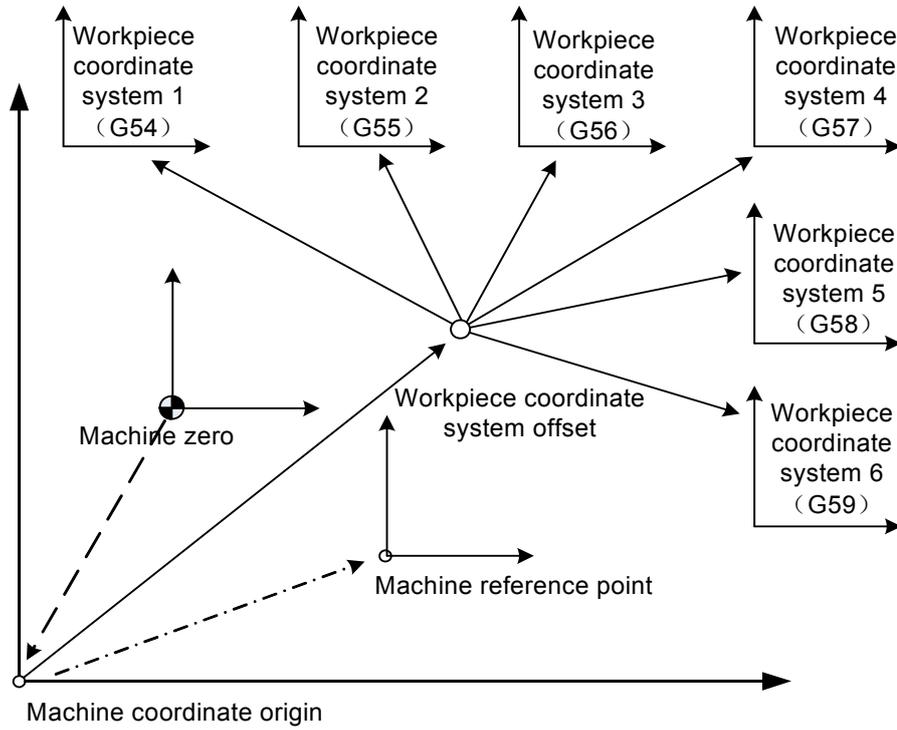
5. The external workpiece zero offset or workpiece zero offset can be altered by G10, which is shown as follows:

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).

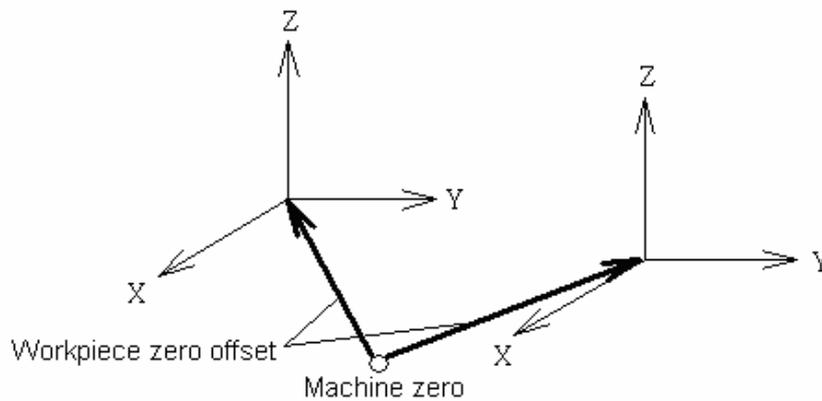
By G10 instruction, each coordinate system can be altered respectively.

**Programming**



**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 P10~13 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 P15~43. 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 positioning to a position (X=100, Y=20) in G55 workpiece coordinate system.

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

### 4.2.9 Additional workpiece coordinate system

Except 6 workpiece coordinate systems (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.

When the P address of the additional workpiece coordinate and the other instructions containing P address are in a same block, they share this P address together.

### 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 only 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. 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.

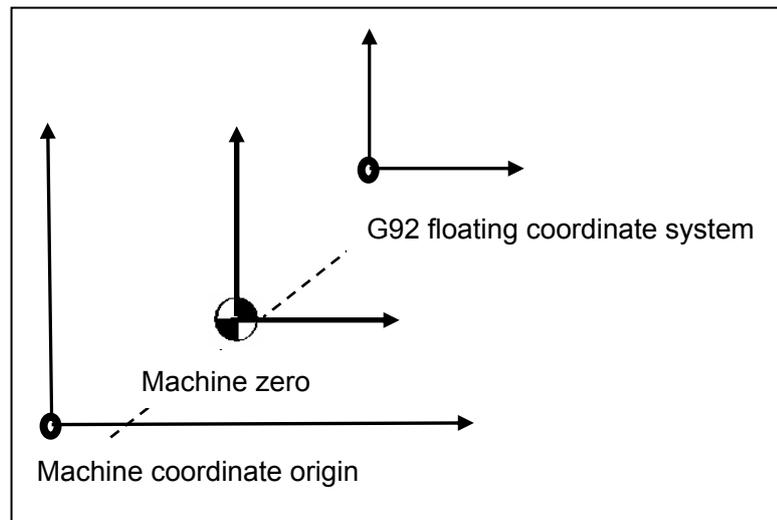
Note: when G53 is specified, the tool radius compensation and tool length offset are cancelled temporarily and they will be restored in the next compensation axis block.

**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 Fig. 4-2-11-1 shows, the origin of the G92 floating coordinate system is the value in machine coordinate system, which is irrelevant to the workpiece coordinate system,

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 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:

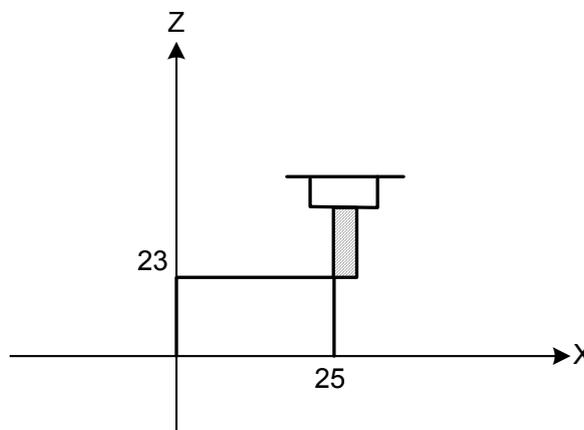


Fig. 4-2-11-2

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

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

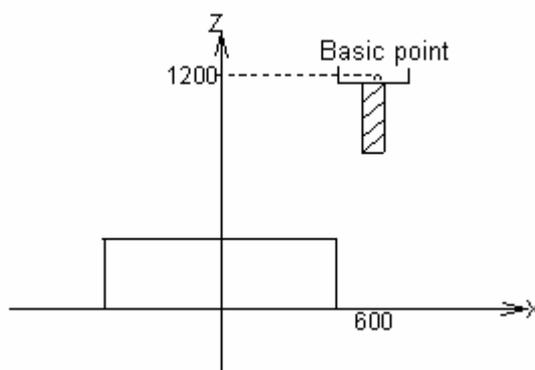


Fig. 4-2-11-3

As Fig. 4-2-11-3 shows, specify the workpiece coordinate system by block "G92 X600 Z1200"(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.

#### 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.

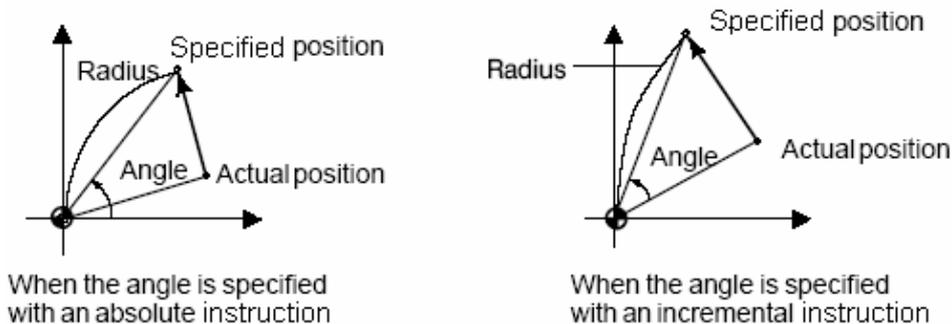
If G16 is set, the coordinate can be input by polar coordinate radius and angle. The positive of angle is the CCW direction of the 1<sup>st</sup> axis positive direction in a plane selected; while the negative is CW direction. Both the radius and angle can use the absolute or incremental instructions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

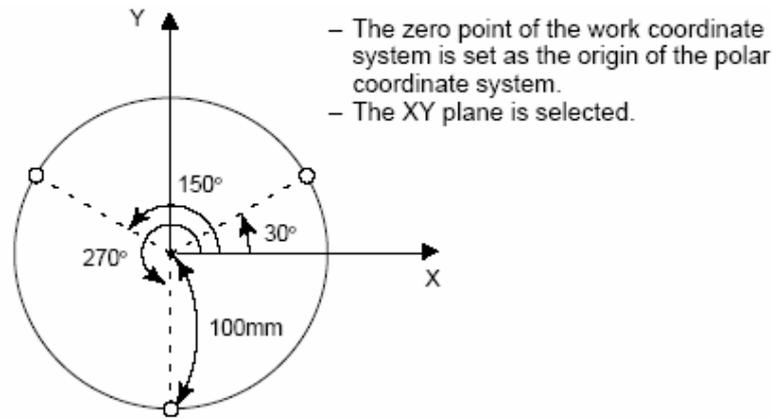
- 1 In G90 absolute mode, if G16 is specified, the workpiece coordinate system origin is regarded as the polar coordinate system origin.



**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.

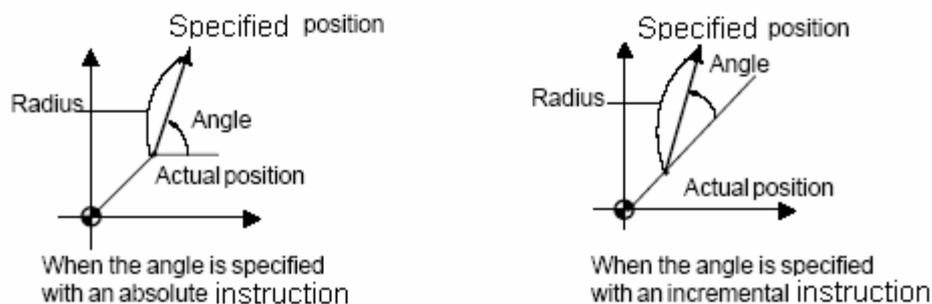
Example: Bolt hole circle (the workpiece coordinate system zero point is set as the polar coordinate system origin, selecting X-Y plane)



**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 Y30 Z-20 R -5 F200; 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  
 X100 Y30 Z-20 R -5 F200; 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 (otherwise, unexpected state will occurred to cause workpiece damage and person hurt.)

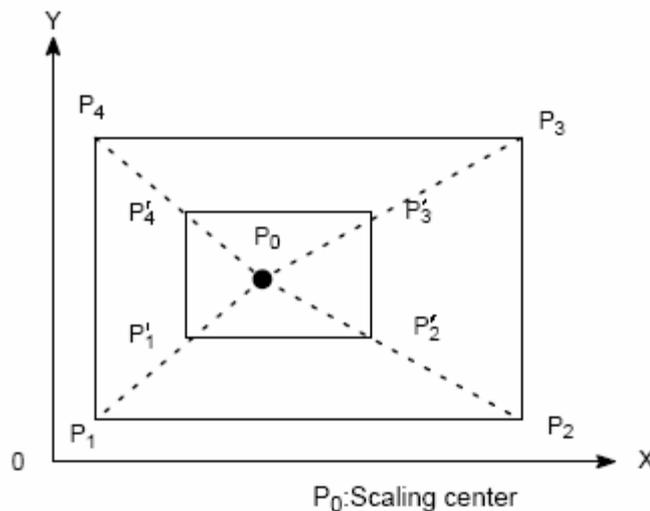


Fig. 4-2-14-1 Scaling (P1'P2'P3'P4 → P1'P2'P3'P4 )

**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

**Programming**

are not specified, the tool current position will be specified for 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, 4<sup>TH</sup> and 5<sup>th</sup> 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 X axis scaling is set by bit parameter NO: 47#3, the effectiveness of the Y axis scaling is set by bit parameter NO: 47#4, the effectiveness of the Z axis scaling is set by bit parameter NO: 47#5, and the scaling ratio of each axis is set by bit parameter NO: 47#6. (0: instructed with P, 1: instructed with I, J, K.)
- 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)

```
G90 G0 X0 Y100;
G51 X0 Y0 Z0 I2 J1;
G02 X100 Y0 R100 F500;
```

Above commands are equivalent to the following one:

```
G90 G0 X0 Y0 Z0;
G02 X200 Y0 R200 F500;
```

Scaling of radius R depends on I or J which is larger

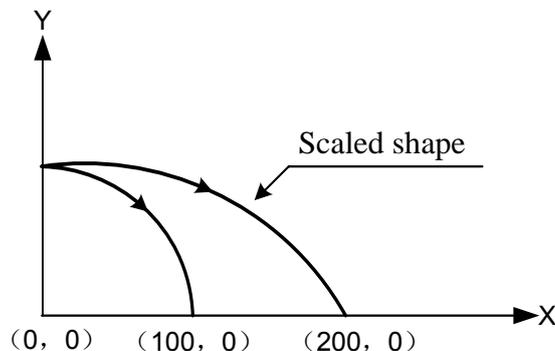
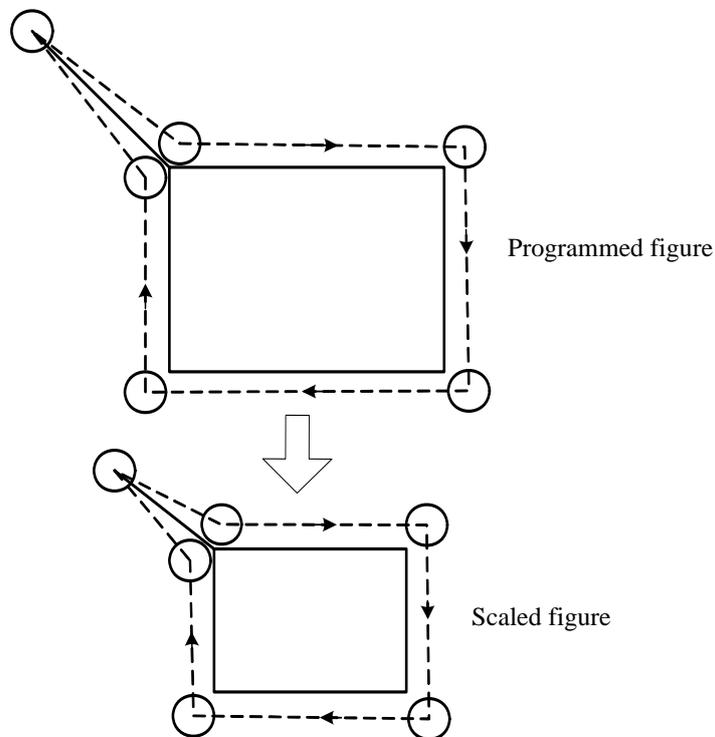


Fig. 4-2-14-2 Scaling of circular interpolation 1

If the axes scaling ratio are different, and the circular interpolation is programmed by I, J, K. if the arc is failed, alarm for it occurs by the system.

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



The tool radius compensation values are not scaled

Fig. 4-2-14-3 Scaling of tool radius compensation

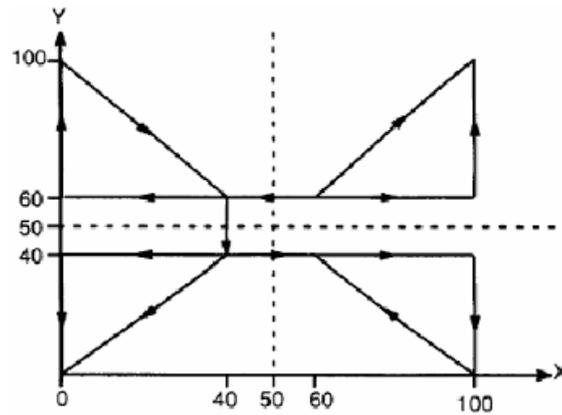
Example for mirror image program:

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;
```

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, R value and P value 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 MANUAL 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 following:**

- 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 comprises 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.

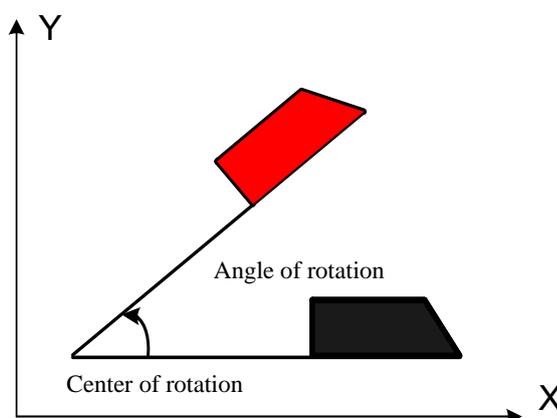


Fig. 4-2-15-1

**Explanation:**

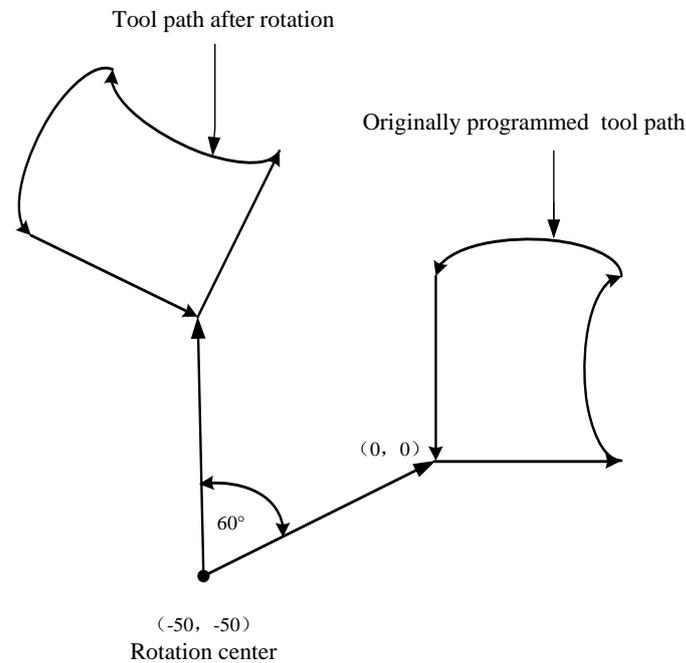
- 1 G68 is an optional parameter with 2 positioning parameters that are used for specifying the rotation center. If the rotation center is not specified, the tool current position is regarded as the center by system. The positioning parameters are relevant to the current coordinate plane, while X, Y 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;
M30
```

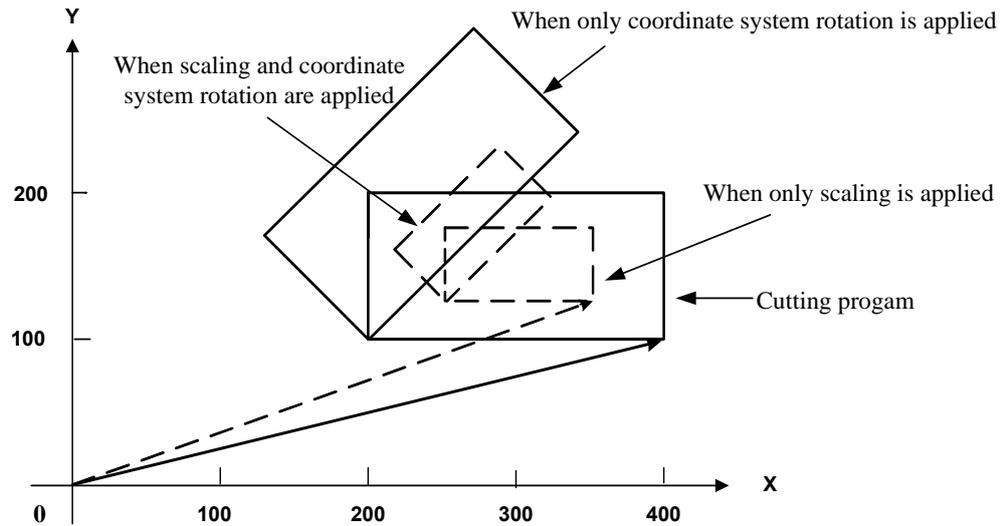


Fig. 4-2-15-3

Example 3 : Repetition of G68

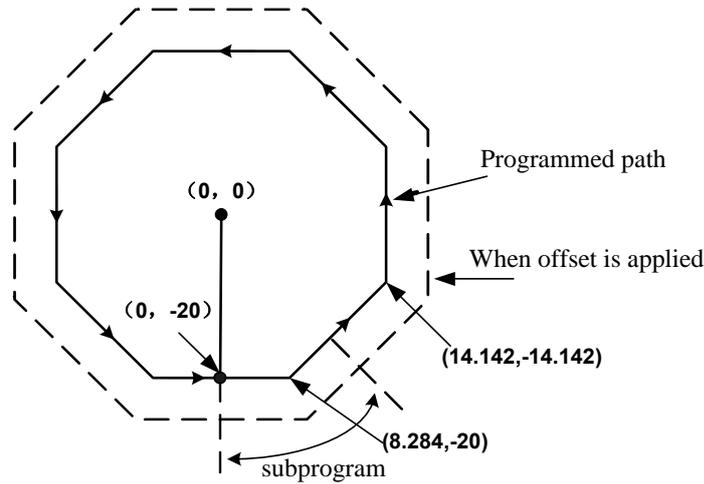
By program (main program)

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

Subprogram 2200

```
G91
G68 X0 Y0 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\_

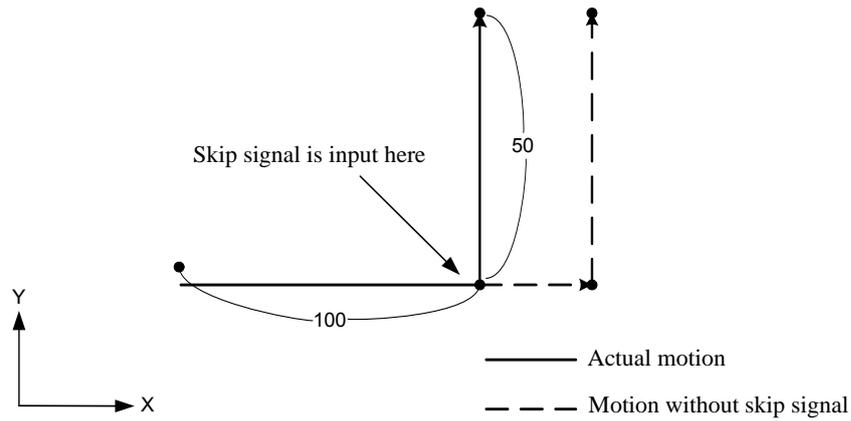
**Function:** The linear interpolation can be specified like G01 after G31 instruction. During the execution of G31, the current instruction execution will be interrupted to execute next block if an external skip signal is entered. While the working end point is specified not by programming but by signals 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-mode 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.

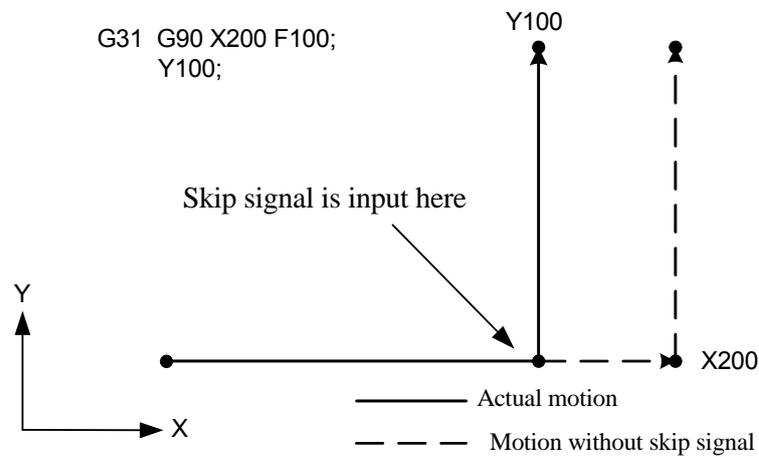
**Example:**

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



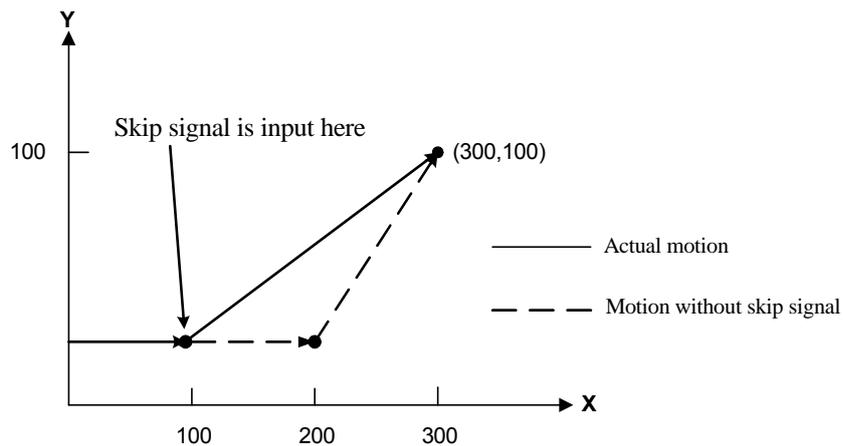
**Fig. 4-2-16-1 A single axis moving specified by incremental values of next block**

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: Program skip should not be supported in DNC and MDI single mode.**

#### **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:**

- 1 This function must be specified by a single block at the beginning of the program before the coordinate system setup.
- 2 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 **The tool compensation value must be preset by the minimum incremental input unit when inch system is converted to metric system or the reverse.**
- 2 **For the 1<sup>st</sup> G28 instruction, the running from the intermediate point is the same as the MANUAL reference point return when inch system is converted to metric system or the reverse.**
- 3 **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 .**
- 4 **The inch/metric system for program input can be set by bit parameter NO:00#2.**
- 5 **The inch/metric system for program output can be set by bit parameter NO:03#0.**
- 6 G20 or G21 must be specified in single block.

#### **4.2.18 Optional 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、 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:

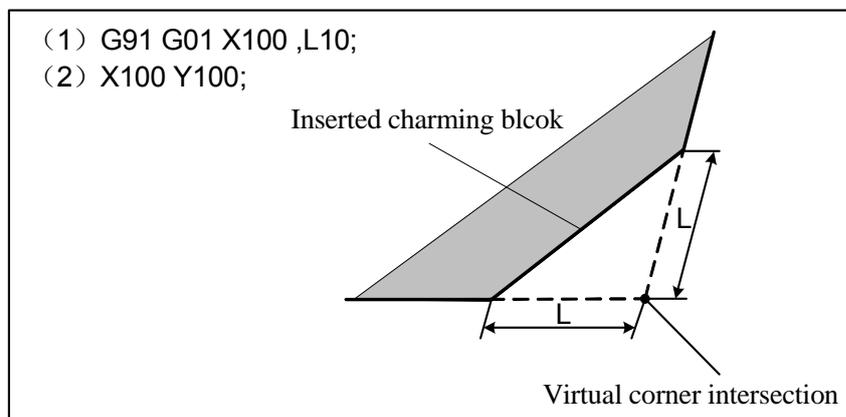


Fig. 4-2-18-1

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

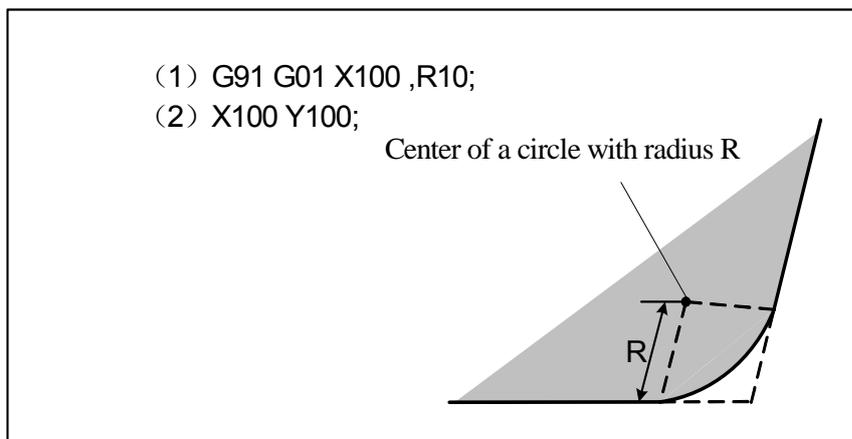


Fig. 4-2-18-2

**Restriction**

- 1 Chamfering and corner rounding can only be performed in the plane specified, 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 if chamfering and corner rounding value is negative, system adopts the absolute value.

### 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.

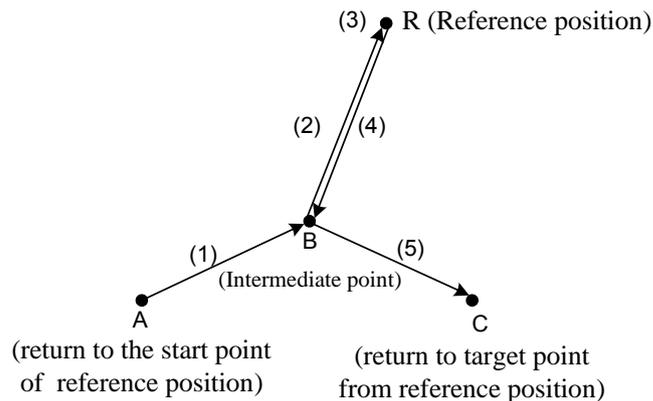


Fig. 4-3-1

#### 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.

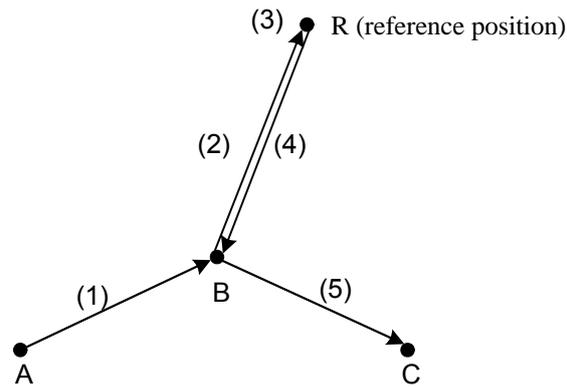


Fig. 4-3-1-1

- 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-mode 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~P48.

### 4.3.2 2nd, 3rd, 4th 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 reference point via the intermediate point specified by G30.

#### 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 P50~63 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 check for returning to reference point, which code is specified by X\_ Y\_ Z\_.

#### 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.

- 4 The X, Y, Z coordinate point position specified by G27 is a position in machine coordinate system.

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:**

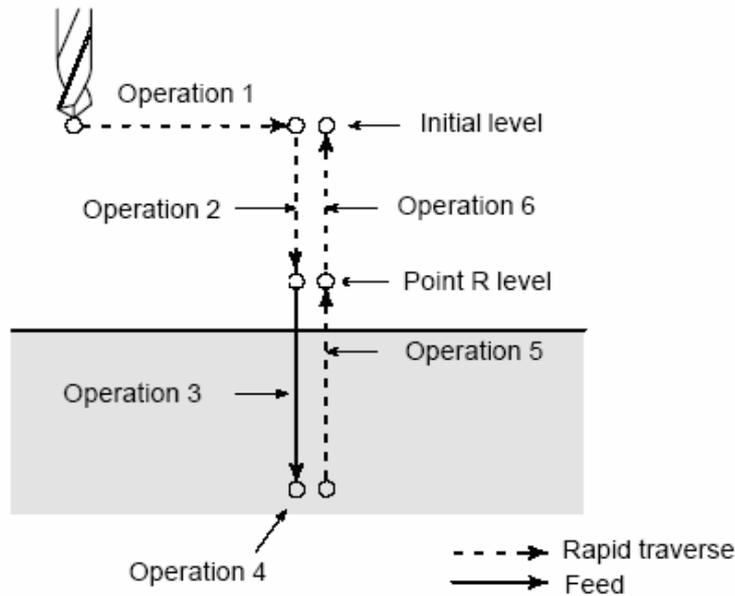
- 1、 G27 instruction positions the tool at a traverse speed. If the tool reaches the reference point, the reference point return indicator lights up. However, if the position reached by the tool is not the reference point, an alarm is issued.
- 2、 In machine lock mode, even G27 is specified and the tool 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.

### 4.4 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

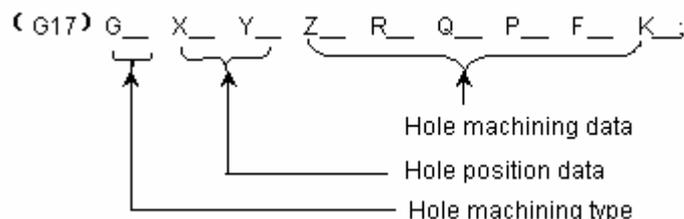
**Initial level and R level**

**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.

**Z, R:** The hole bottom parameter Z and R must be correctly specified simultaneously while performing the 1<sup>st</sup> drilling operation(omitting unallowable) or just modal is changed, Z axis operation is invalid..

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  |

**Programming**

|  |  |  |
|--|--|--|
|  |  | 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 mode 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 In canned cycle, scale Z axis (cutting axial ) is ineffective.
- 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:

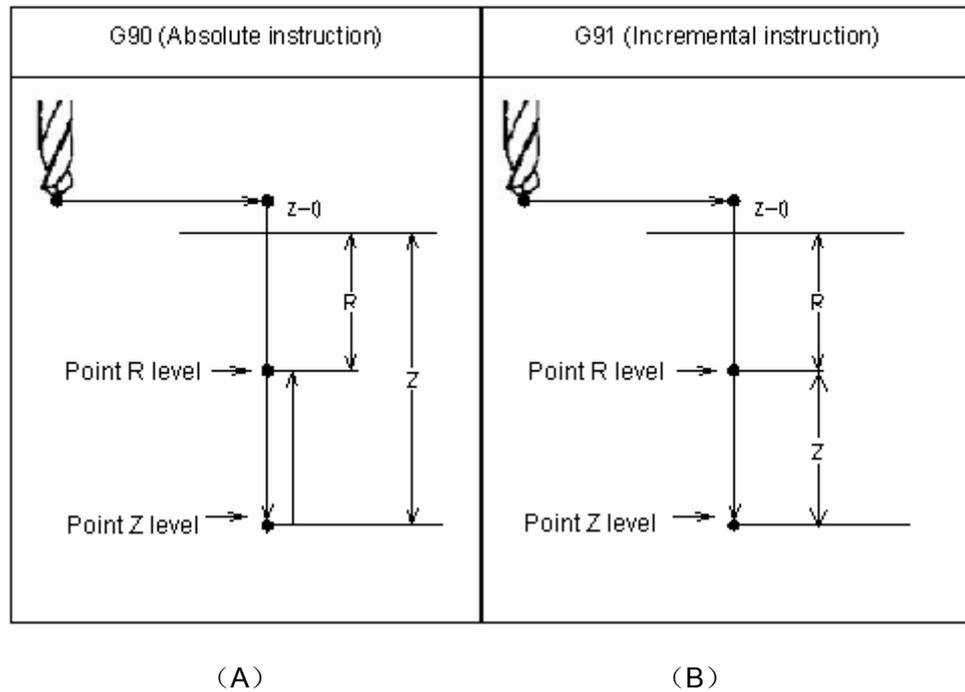
**Table 4-4-2**

| 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 deal with according to the negative/positive sign of

them.)



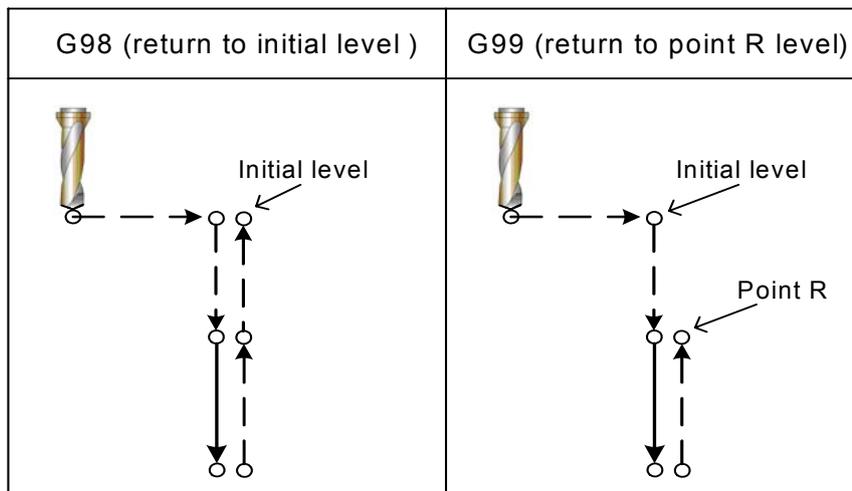
**Fig. 4-4-2**

**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.

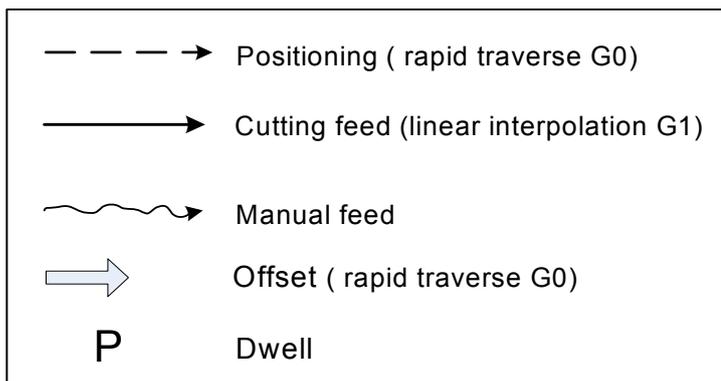
G98 is the system default mode.



**Fig. 4-4-3**

The following symbols are used for the canned cycle illustration:

**Programming**



**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                                     |
|------------|-------------------------|------------------------------|--------------------------|---|
| <b>G22</b> | <b>Feed</b>             |                              | Rapid                    | CCW inner circular groove rough milling         |
| <b>G23</b> | <b>Feed</b>             |                              | Rapid                    | CW inner circular groove rough milling          |
| <b>G24</b> | <b>Feed</b>             |                              | Rapid                    | CCW finish-milling within a circle cycle        |
| <b>G25</b> | <b>Feed</b>             |                              | Rapid                    | CW finish-milling within a circle cycle         |
| <b>G26</b> | <b>Feed</b>             |                              | Rapid                    | CCW outer circle finish-milling cycle           |
| <b>G32</b> | <b>Feed</b>             |                              | Rapid                    | CW outer circle finish-milling cycle            |
| <b>G33</b> | <b>Feed</b>             |                              | Rapid                    | CCW rectangle groove rough milling              |
| <b>G34</b> | <b>Feed</b>             |                              | Rapid                    | CW rectangle groove rough milling               |
| <b>G35</b> | <b>Feed</b>             |                              | Rapid                    | CCW inner rectangle groove finish-milling cycle |
| <b>G36</b> | <b>Feed</b>             |                              | Rapid                    | CW inner rectangle groove finish-milling cycle  |
| <b>G37</b> | <b>Feed</b>             |                              | Rapid                    | CCW rectangle outside finish-milling cycle      |
| <b>G38</b> | <b>Feed</b>             |                              | Rapid                    | CW rectangle outside finish milling cycle       |
| G73        | Intermittent feed       |                              | Rapid feed               | High-speed peck drilling cycle                  |
| G74        | Feed                    | Dwell→spindle CW             | Feed                     | Counter tapping cycle                           |
| G76        | Feed                    | Oriented spindle stop        | Rapid feed               | Fine boring                                     |
| G80        |                         |                              |                          | Cancel  |
| G81        | Feed                    |                              | Rapid feed               | Drilling, spot drilling                         |
| G82        | Feed                    | Stop                         | Rapid feed               | Drilling, counterboring                         |
| G83        | Intermittent feed       |                              | Rapid feed               | Peck drilling cycle                             |

|     |      |                        |            |         |
|-----|------|------------------------|------------|---------|
| G84 | Feed | Dwell →<br>spindle CCW | Feed       | Tapping |
| G85 | Feed |                        | Feed       | Boring  |
| G86 | Feed | Spindle stop           | Rapid feed | Boring  |
| G87 | Feed | Spindle CCW            | Rapid feed | Boring  |
| G88 | Feed | Dwell →<br>spindle CCW | MANUAL     | Boring  |
| G89 | Feed | Dwell                  | Feed       | Boring  |

**Restriction**

In canned cycle, 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:**

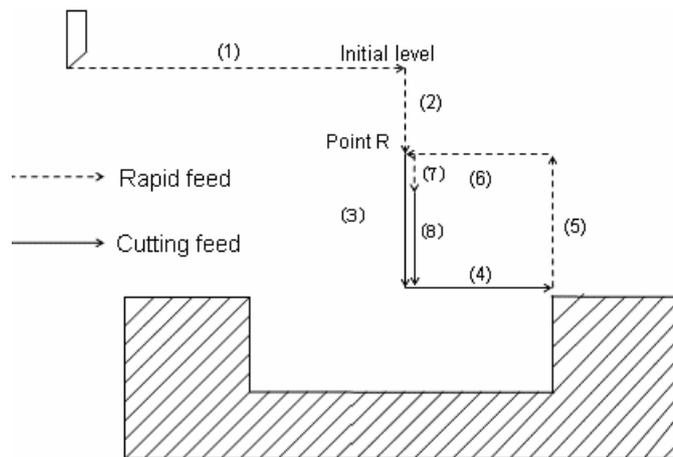
- G22: CCW inner circular groove rough milling
- G23: CW inner circular groove rough milling
- X、Y: The start point within X, Y plane
- Z: Machining depth, which is absolute position in G90 and position to R reference level in G91
- R: R reference level, which is absolute position in G90 and position to start point of this block in G91
- I: 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 diameter number, ranging within 1 ~ 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 federate and then infeed to circle center;
- (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 positionign to the starting point;
- (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

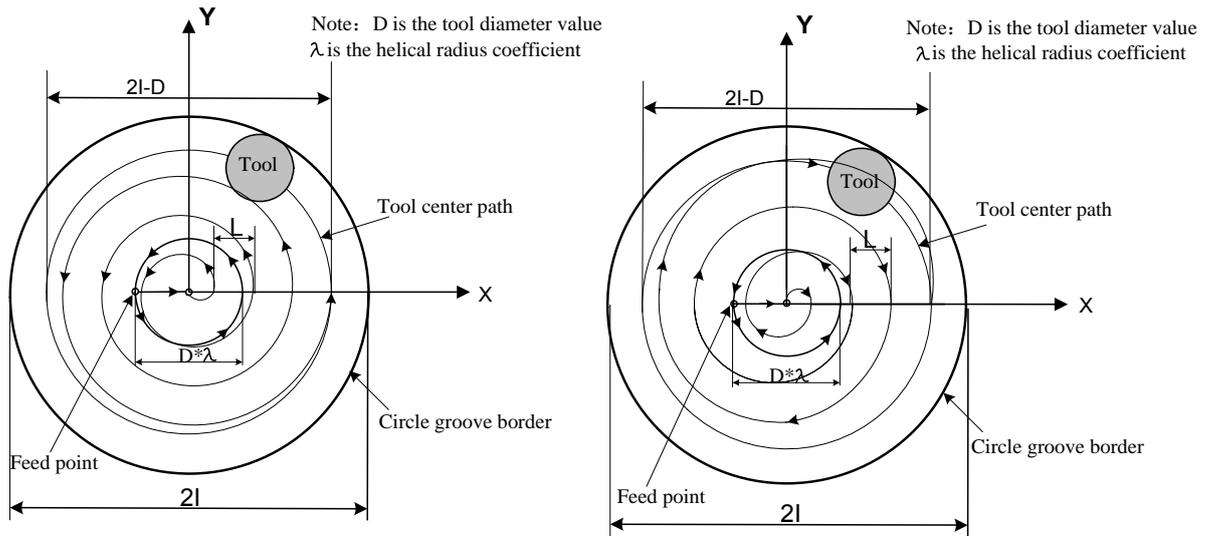


Fig. 4-4-1-2

Note:

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

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

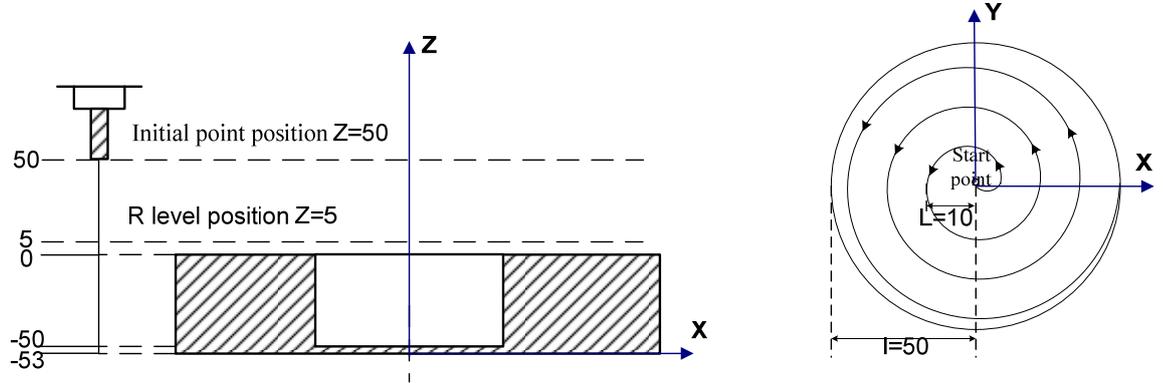


Fig. 4-4-1-3

G90 G00 X50 Y50 Z50; (G00 rapid positioning)

G99 G22 X25 Y25 Z-50 R5 I50 L10 W20 Q10 V10 D1 F800;

(Groove rough milling cycle within a circle)

G80 X50 Y50 Z50;

(Canned cycle cancel and return from R level)

M30;

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 positioning in canned cycle is ignored.

#### 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 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: Milling circle radius, ranging within 0.0001mm~99999.9999mm, 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 diameter number, ranging within 1 ~256. D0 is defaulted for 0. The tool diameter value is obtained by the given number.

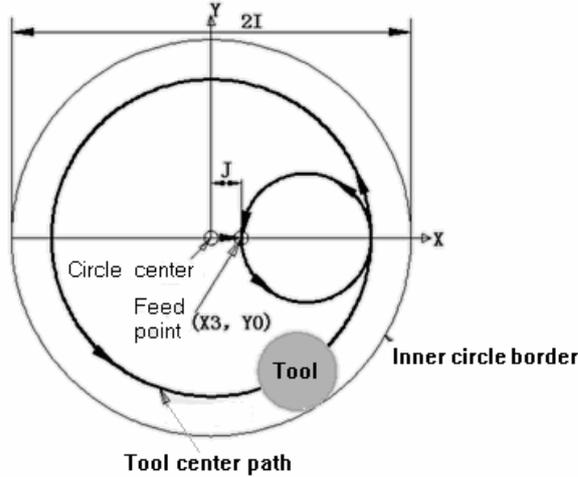
K: Repetitions

**Cycle process:**

- (1) Rapid to a location within XY plane;
- (2) Rapid down to R level;
- (3) Feed to the machining start point of hole bottom;
- (4) To position to the start point from current position at the 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:**

G24: CCW finish-milling within a circle cycle



G25: CW finish-milling within a circle cycle

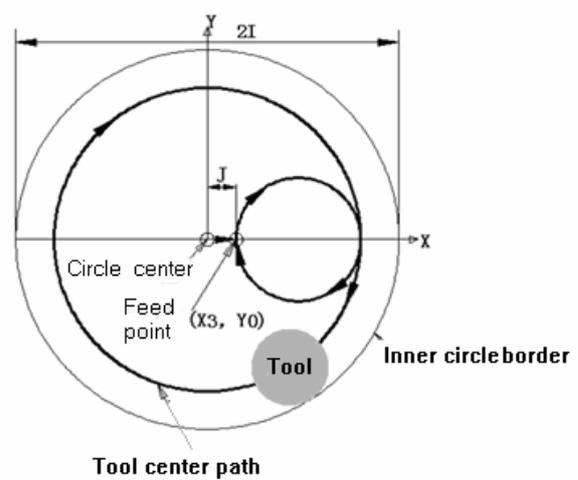


Fig. 4-4-2-1

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

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

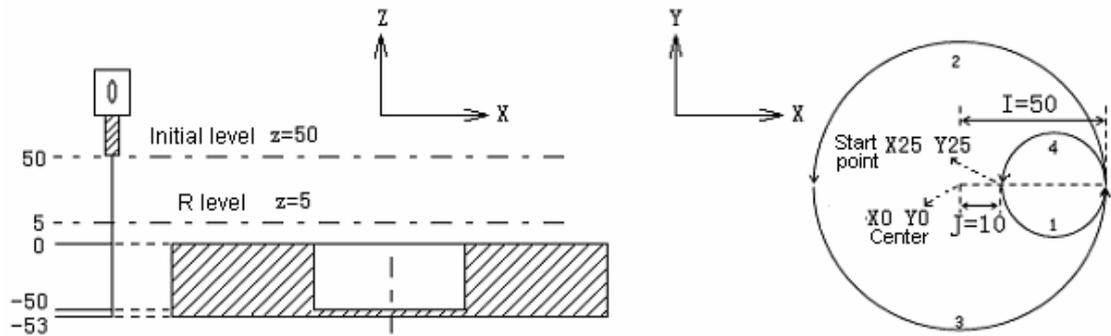


Fig. 4-4-2-2

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;

Cancellation: G codes in 01 group (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 positioning 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 mill 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 0.0001mm~99999.9999mm, use the absolute value if it is a negative one.

J: Distance from the milling start point to milling circle center, ranging within 0.0001mm~99999.9999mm, use the absolute value if it is a negative one

D: Tool radius number, ranging within 1 ~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 interpolate by the transition arc 1 from the start point;
- (5) To make circular interpolation for the whole circle by arc 2, arc 3
- (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:**

G26: CCW outer circle finish milling cycle

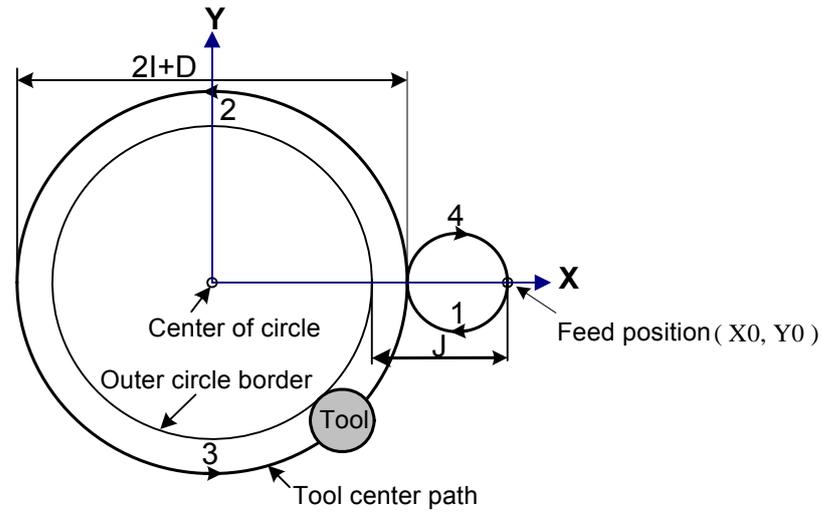


Fig. 4-4-3-1

G32: CW outer circle finish milling cycle

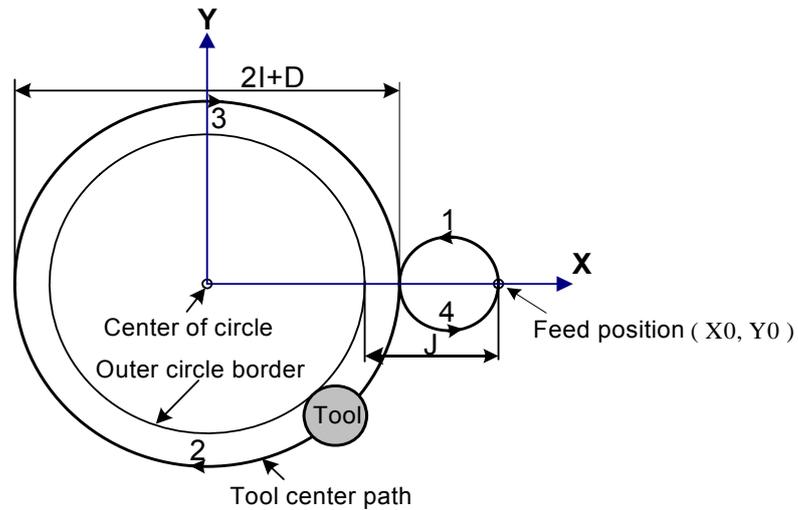
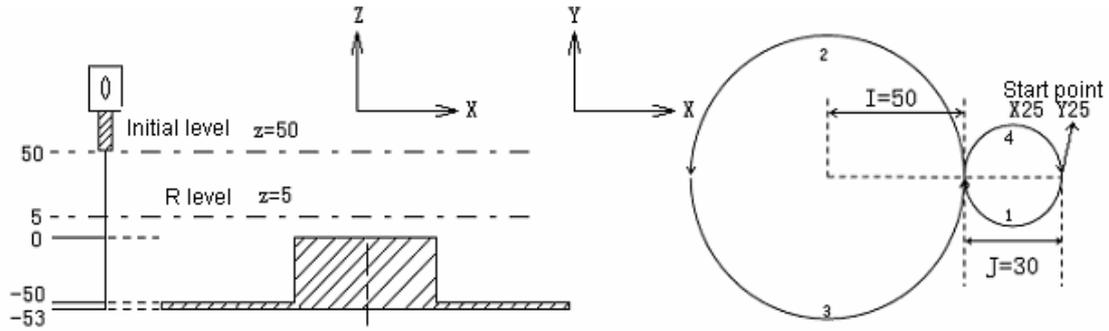


Fig. 4-4-3-2

**Explanation:**

In outer circle finish milling, the interpolation directions of transition arc and finish milling arc are different, while 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. 4-4-3-3**

```
G90 G00 X50 Y50 Z50;           (G00 rapid positioning)
G99 G26 X25 Y25 Z-50 R5 I50 J30 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;
```

Cancellation: G codes in 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 .

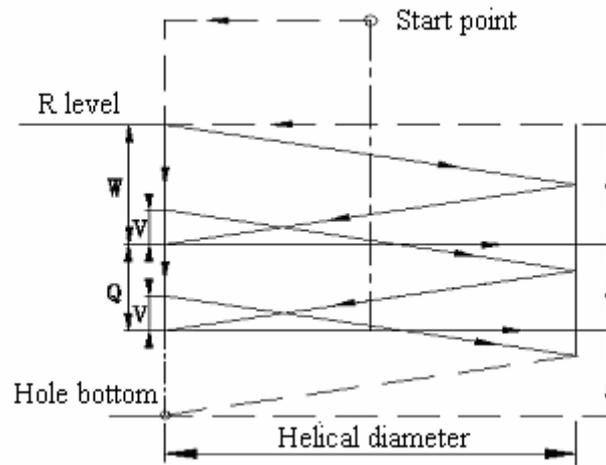
- J: Rectangular groove width in Y axis, which should be over the tool radius and helical feed radius should be less than half of it .
- L: Cutting width increment within a specified plane, which should be less than the tool diameter and over 0
- W: Initial cut depth in Z axis, which is a downward distance from R level and is over 0 (if the initial cut exceeds the groove bottom, it will cut at the bottom position)
- Q: Cut 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 diameter number, ranging within 1 ~ 256, D0 is defaulted for 0. The current tool diameter value is given by the number specified.
- K: Repetitions

**Cycle process:**

- (1) Rapid to a start point of helical feed 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 N0. 269 value
- (4) Feed to the rectangle center;
- (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) ~ (9) 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.

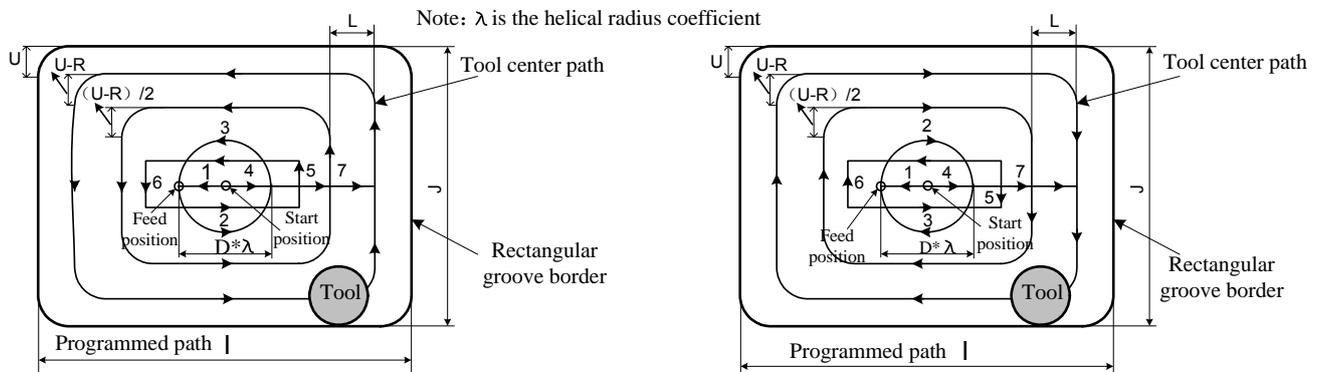
**Instruction path:**

**Programming**



G33 CCW rectangular groove rough milling

G34 CW rectangular groove rough milling

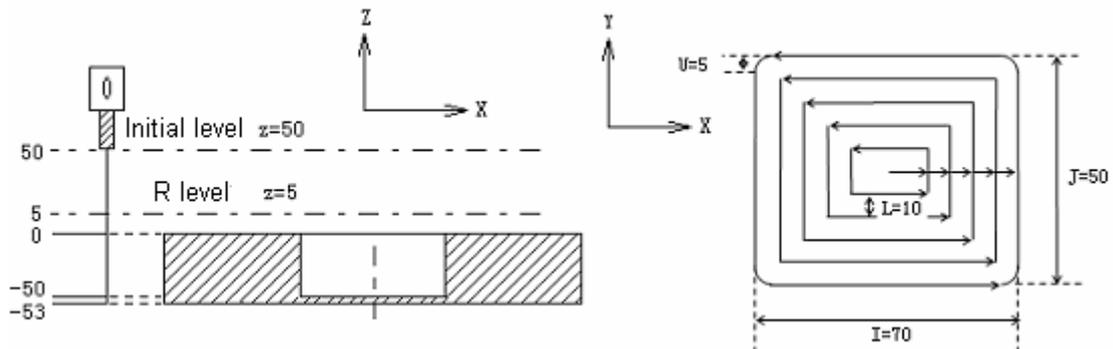


**Fig. 4-4-4-1**

**Note:** The NO:12#1 should be set to 1 when this 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)



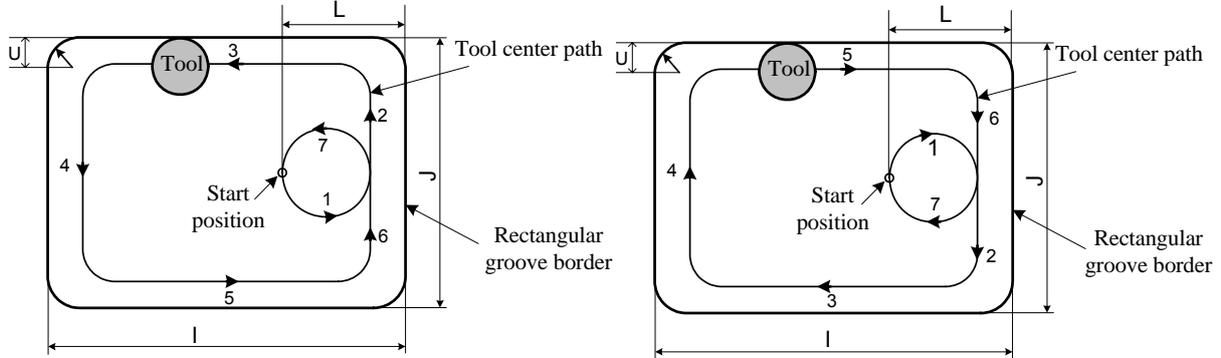
point;

(7) Return to the initial level or R level according to G98 or G99 instruction.

**Instruction path:**

G35: CCW rectangular groove finish milling cycle

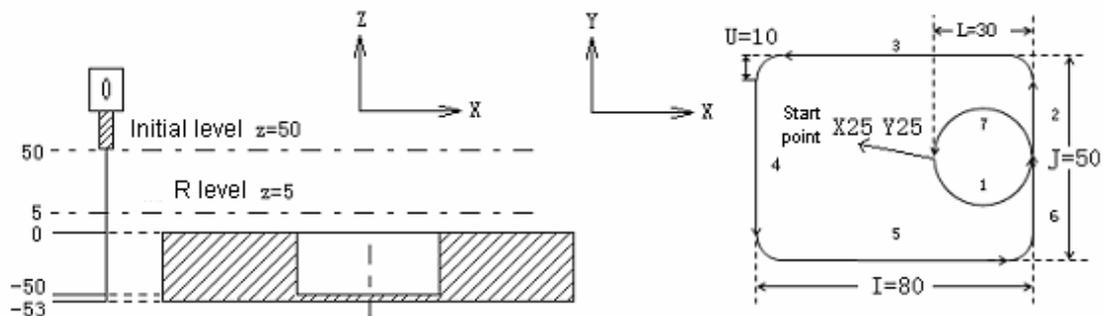
G36: CW rectangular groove finish milling cycle



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

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

**Example:** To finish mill a 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 Y0 Z-50 R5 I80 J50 L30 U10 D1 F800; (Canned cycle starts, and go down to the bottom to perform the rectangular groove finish milling)

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

M30;

**Cancellation:** G codes in 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 positioning 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

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 0.0001mm ~99999.9999mm

J: Rectangular width in Y axis, ranging within 0.0001mm ~99999.9999mm

L: Distance of start point to rectangular side in X axis, ranging within 0.0001mm ~99999.9999mm

U: Corner radius, no corner transition if omitted

D: Tool diameter number, ranging within 1 ~ 256, D0 is defaulted for 0. The current tool diameter value is given by the number specified

K: Repetitions

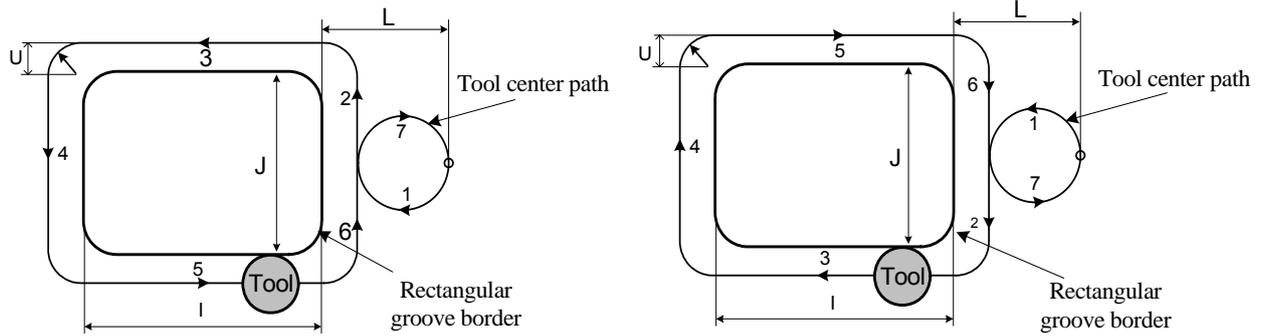
**Cycle process:**

- (1) Rapid to start point of XY plane;
- (2) Rapid down to R level;
- (3) Feed to the hole bottom;
- (4) To make circular interpolation by the transition arc 1 from the start point;
- (5) To make linear and circular interpolation by the path 2-3-4-5-6;
- (6) To make circular interpolation by the path of transition arc 7 and return to the start point;
- (7) 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:** To finish mill outer rectangle by canned cycle G37 instruction:

```
G90 G00 X50 Y50 Z50; (G00 rapid positioning)
```

```
G99 G37 X25 Y25 Z-50 R5 I80 J50 L30 U10 D1 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 in 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 in 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\_: Repetitions

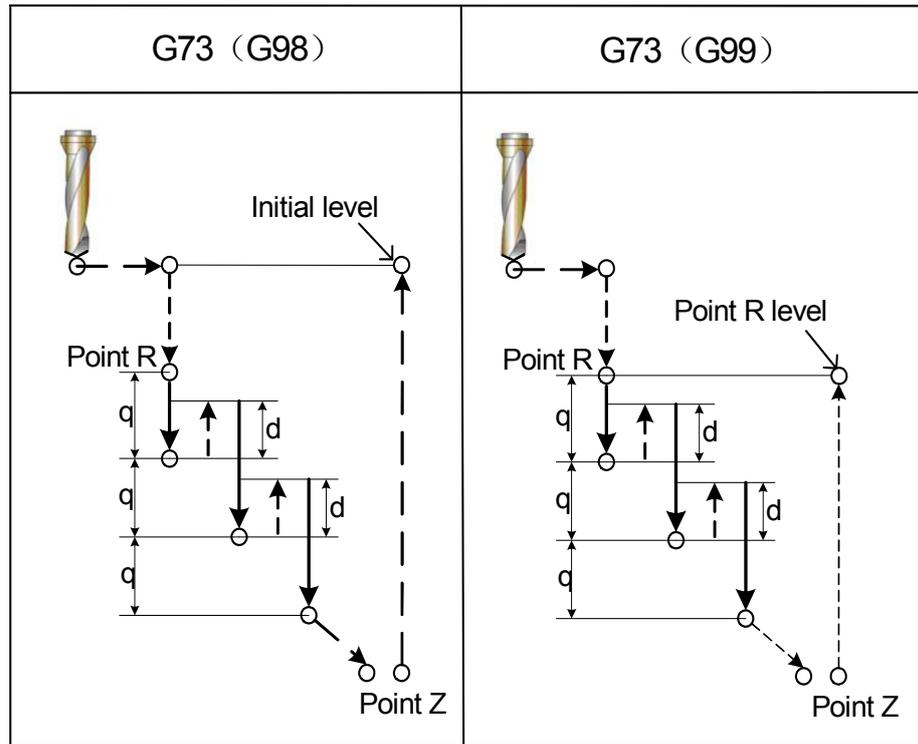


Fig. 4-4-7-1

**Z, R:** The hole bottom parameter Z and R must be correctly specified while performing the 1<sup>st</sup> drilling operation (omitting unallowable) or the alarm is issued.

**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.4-4-1-1) 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.

**Note 1** In current version, M00, M01, M02, M06, M30, M98, M99 will be executed after current statement is executed.

**Note 2** If parameter Q is not specified or is set to be 0, hole positioning in XY plane is executed but hole drilling not. If Q value is negative, system will execute intermittence feed at its absolute value.

**Note3** Tool length compensation: if the tool length compensation (G43, G44 or G49) and canned cycle command are in a same block, the offset value is either added or cancelled while positioning to point R level; if the tool length compensation (G43, G44 or G49) is specified in a block alone, the offset value is either added or cancelled realtime.

Cancellation: G codes in 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 in 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\_: Repetitions (if necessary)

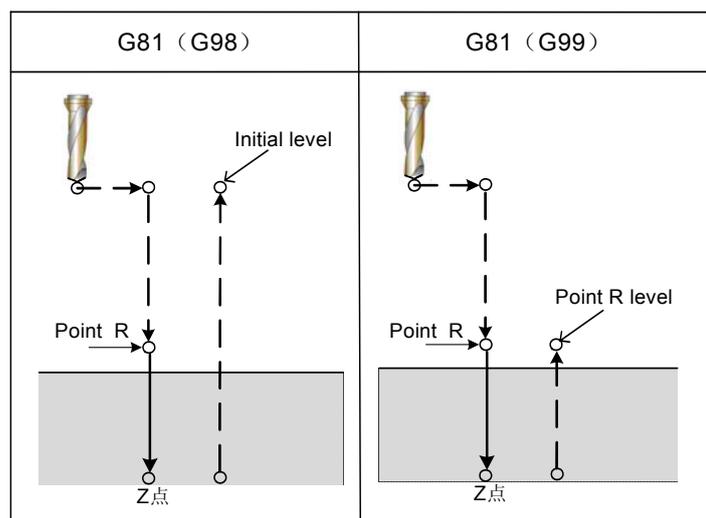


Fig. 4-4-8-1

**Z, R:** The hole bottom parameter Z and R must be correctly specified simultaneously while performing the 1<sup>st</sup> drilling operation(omitting unallowable) or just modal is changed, Z axis operation is invalid..

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.

**Note 1** In current version, M00, M01, M02, M06, M30, M98, M99 will be executed after current statement is executed.

**Tool length compensation:** if the tool length compensation (G43, G44 or G49) and canned cycle command are in a same block, the offset value is either added or cancelled while positioning to point R level; if if the tool length compensation (G43, G44 or G49) is specified ina block alone, the offset value is either added or cancelled realtime..

**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

### Programming

---

|                    |  |
|--------------------|--|
| Y-550.;            | Positioning, drilling 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;               |  |

Cancellation: G codes in 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 in positioning in canned cycle is ignored.

#### 4.4.9 Drilling cycle, counterboring 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\_: Repetitions

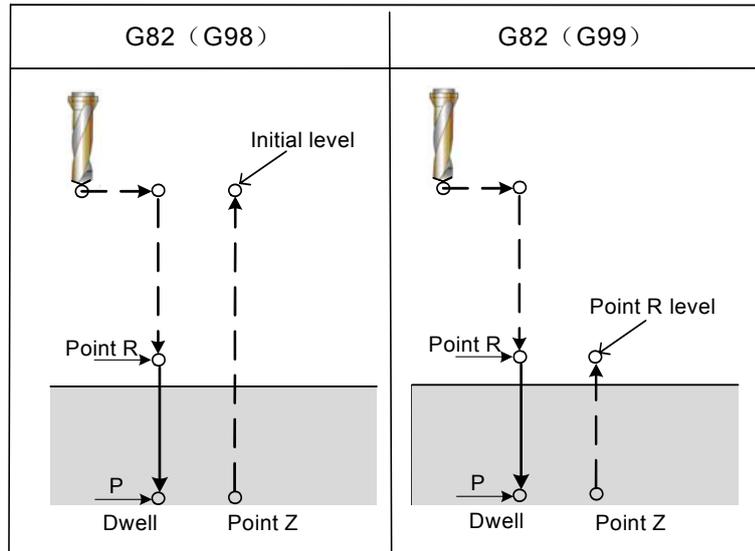


Fig. 4-4-9-1

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.

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.

**Note 1** In current version, M00, M01, M02, M06, M30, M98, M99 will be executed after current statement is executed.

**Tool length compensation:** if the tool length compensation (G43, G44 or G49) and canned cycle command are in a same block, the offset value is either added or cancelled while positioning to point R level; if if the tool length compensation (G43, G44 or G49) is specified in a block alone, the offset value is either added or cancelled realtime..

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;

Cancellation: G codes in 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 in 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\_: Repetitions

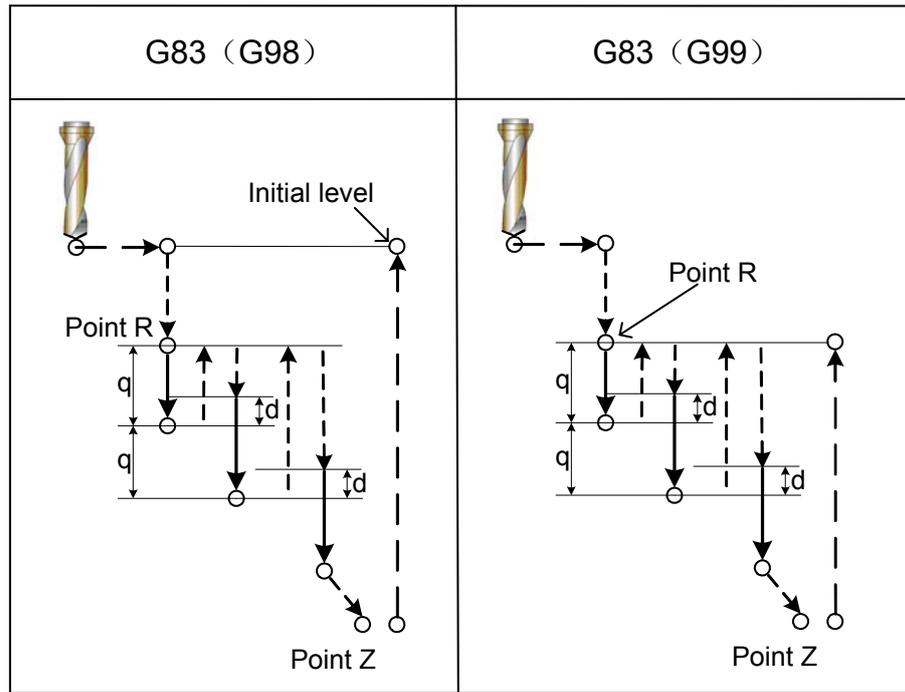


Fig. 4-4-10-1

**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 P271, 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 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.

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.

**Note 1** In current version, M00, M01, M02, M06, M30, M98, M99 will be executed after current statement is executed.

**Tool length compensation:** if the tool length compensation (G43, G44 or G49) and canned cycle command are in a same block, the offset value is either added or cancelled while positioning to point R level; if if the tool length compensation (G43, G44 or G49) is specified ina block alone, the offset value is either added or cancelled realtime..



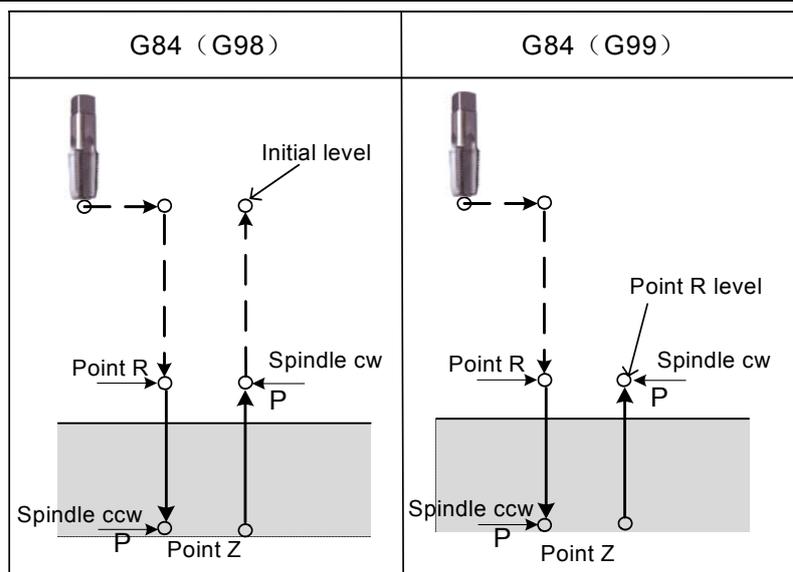


Fig. 4-4-11-1

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.

**Note 1 In current version, M00, M01, M02, M06, M30, M98, M99 will be executed after current statement is executed.**

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 (G43, G44 or G49) and canned cycle command are in a same block, the offset value is either added or cancelled while positioning to point R level; if if the tool length compensation (G43, G44 or G49) is specified ina block alone, the offset value is either added or cancelled realtime..**

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:

**Programming**

S500=500 r /min      F=1.5×500=750mm/min

For multi-start thread, F value can be gotten 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;  |   |

**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 in 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.   |

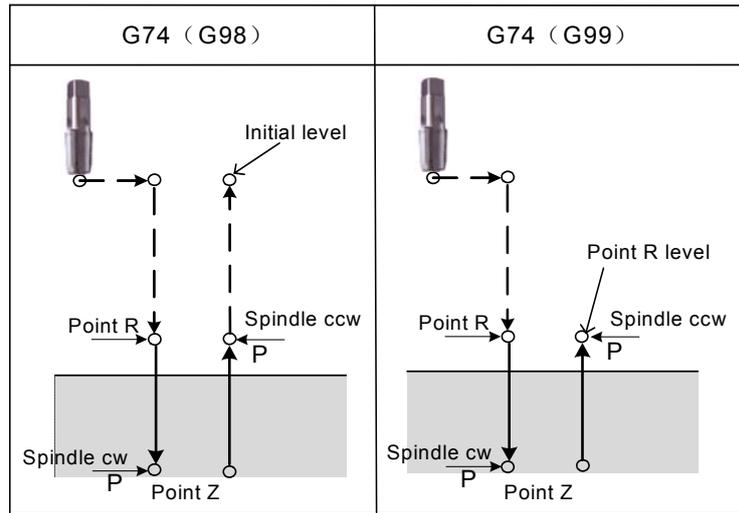


Fig. 4-4-12-1

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.

**Note 1 In current version, M00, M01, M02, M06, M30, M98, M99 will be executed after current statement is executed.**

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 (G43, G44 or G49) and canned cycle command are in a same block, the offset value is either added or cancelled while positioning to point R level; if if the tool length compensation (G43, G44 or G49) is specified ina block alone, the offset value is either added or cancelled realtime..**

**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
    
```

## Programming

---

|                    |   |
|--------------------|---|
| 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;               |   |

**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 in 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   |

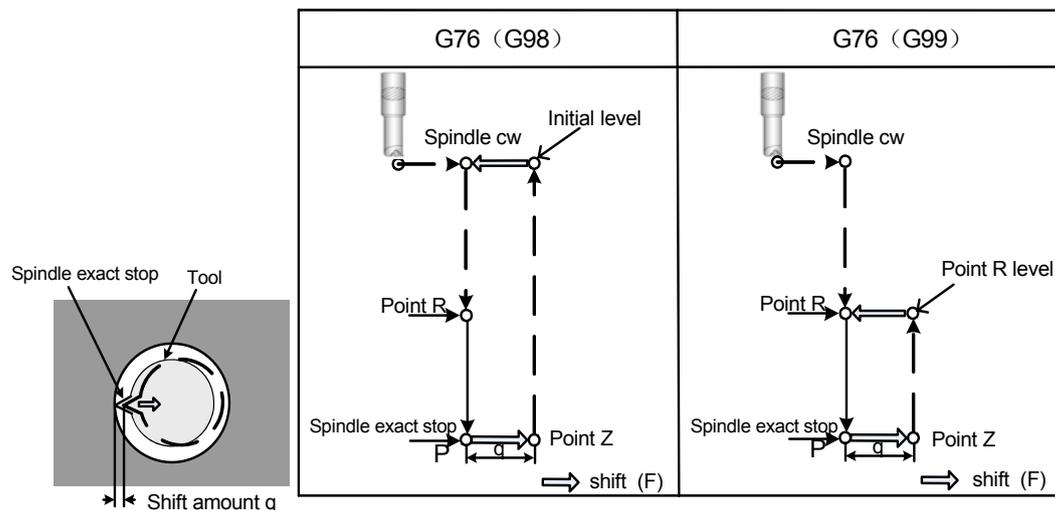


Fig. 4-4-13-1

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.

**Note 1** In current version, M00, M01, M02, M06, M30, M98, M99 will be executed after current statement is executed.

**Tool length compensation:** if the tool length compensation (G43, G44 or G49) and canned cycle command are in a same block, the offset value is either added or cancelled while positioning to point R level; if if the tool length compensation (G43, G44 or G49) is specified in a block alone, the offset value is either added or cancelled realtime..

**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  |

**Cancellation:** G codes in 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 in 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:**

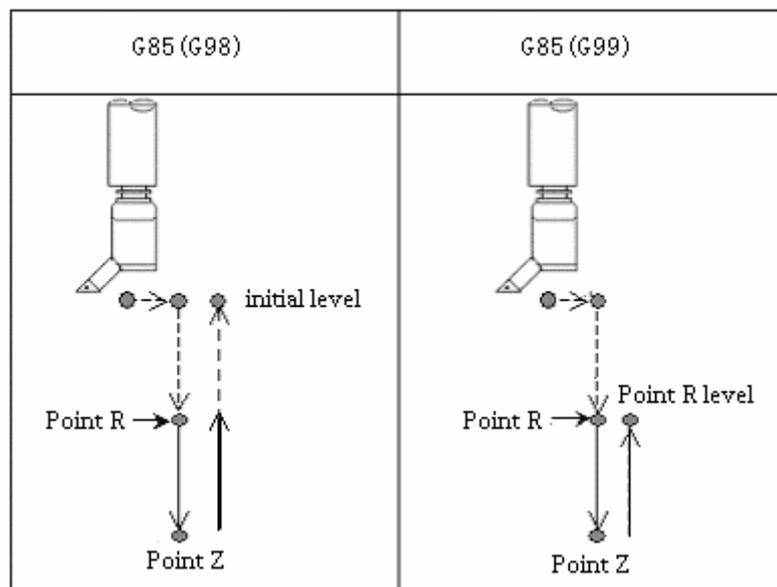
X\_ Y\_: Hole positioning data

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.

**Note 1** In current version, M00, M01, M02, M06, M30, M98, M99 will be executed after current statement is executed.

**Tool length compensation:** if the tool length compensation (G43, G44 or G49) and canned cycle command are in a same block, the offset value is either added or cancelled while positioning to point R level; if if the tool length compensation (G43, G44 or G49) is specified in a block alone, the offset value is either added or cancelled realtime..

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;   |  |

**Cancellation:** G codes in 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 in positioning in canned cycle is ignored.

**4.4.15 Boring cycle G86**

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

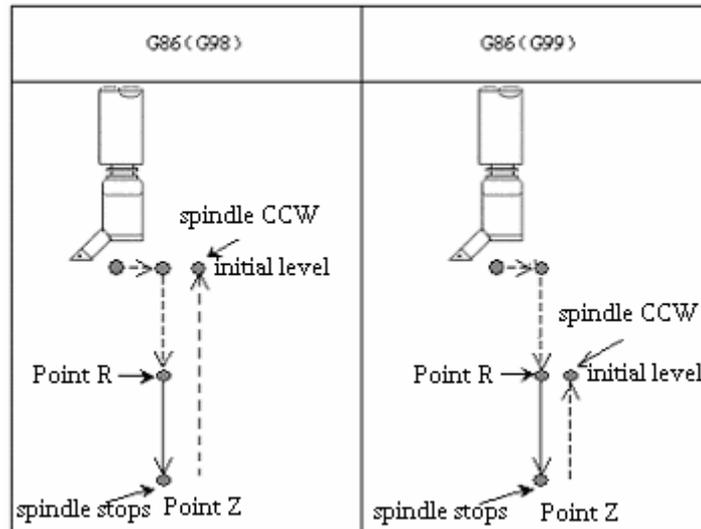
**Function:** It is used to perform a boring cycle.

**Explanation:**

X\_ Y\_: Hole positioning data

**Programming**

- 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\_: Repetitions



**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.

**Note 1** In current version, M00, M01, M02, M06, M30, M98, M99 will be executed after current statement is executed.

**Tool length compensation:** if the tool length compensation (G43, G44 or G49) and canned cycle command are in a same block, the offset value is either added or cancelled while positioning to point R level; if if the tool length compensation (G43, G44 or G49) is specified ina block alone, the offset value is either added or cancelled realtime..

**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;

**Cancellation:** G codes in 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 in 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.

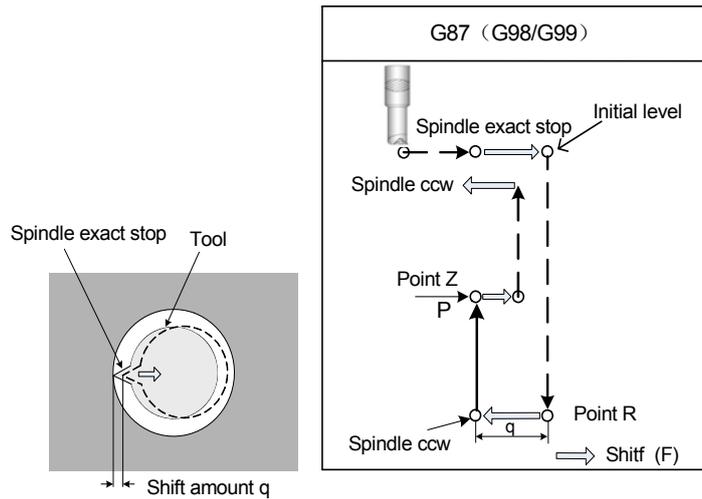
R\_: In incremental programming it specifies the distance from the initial level to point R level; in absolute programming it specifies the absolute coordinate of point R level . (hole bottom)

Q\_: Offset at the hole bottom

P\_: Dwell time

F\_: Cutting feedrate

**Programming**



**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 feed speed 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.

**Note 1** In current version, M00, M01, M02, M06, M30, M98, M99 will be executed after current statement is executed.

**Tool length compensation:** if the tool length compensation (G43, G44 or G49) and canned cycle command are in a same block, the offset value is either added or cancelled while positioning to point R level; if if the tool length compensation (G43, G44 or G49) is specified in a block alone, the offset value is either added or cancelled realtime..

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

**Cancellation:** G codes in 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 in positioning in canned cycle is ignored.

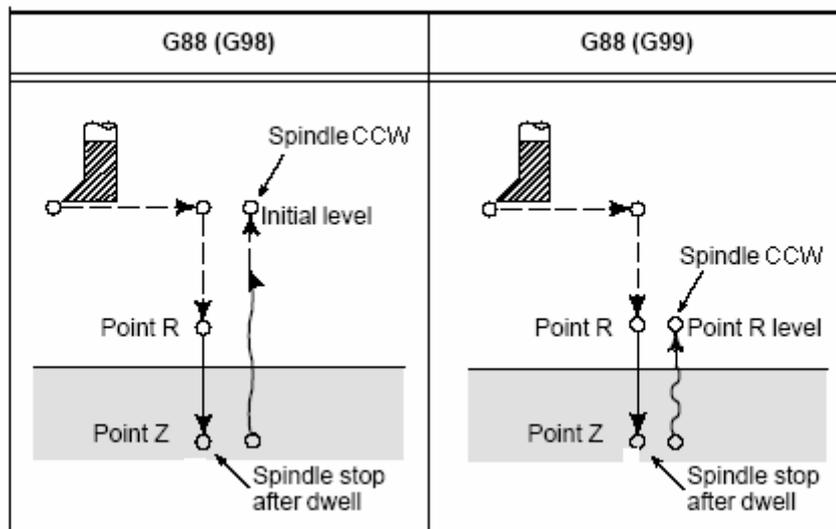
#### 4.4.17 Boring cycle G88

**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.

**Note 1** In current version, **M00, M01, M02, M06, M30, M98, M99** will be executed after current statement is executed.

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 (G43, G44 or G49) and canned cycle command are in a same block, the offset value is either added or cancelled while positioning to point R level; if if the tool length compensation (G43, G44 or G49) is specified ina block alone, the offset value is either added or cancelled realtime..**

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  |

**Cancellation:** G codes in 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 in 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

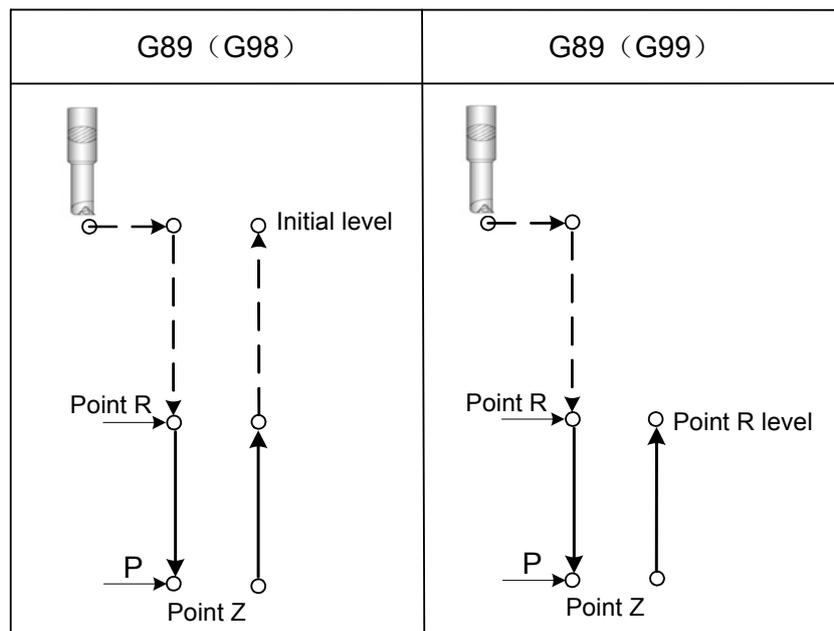


Fig. 4-4-18-1

**Programming**

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.

**Note 1 In current version, M00, M01, M02, M06, M30, M98, M99 will be executed after current statement is executed.**

P is a mode 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 (G43, G44 or G49) and canned cycle command are in a same block, the offset value is either added or cancelled while positioning to point R level; if if the tool length compensation (G43, G44 or G49) is specified ina block alone, the offset value is either added or cancelled realtime..**

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;
    
```

**Cancellation:** G codes in 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 in positioning in 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

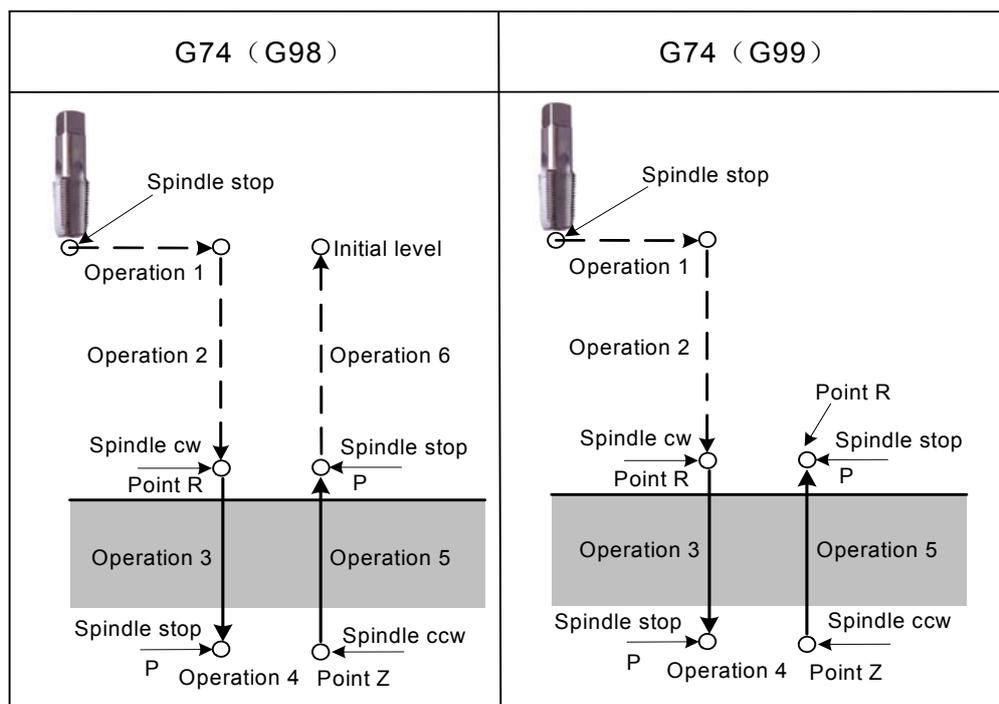
Z\_: In incremental programming it specifies the distance from point R level to the bottom of the hole; in absolute programming it specifies the absolute coordinate of the hole bottom.

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

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.

**Note 1** In current version, M00, M01, M02, M06, M30, M98, M99 will be executed after current statement is executed.

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 (G43, G44 or G49) and canned cycle command are in a same block, the offset value is either added or cancelled while positioning to point R level; if if the tool length compensation (G43, G44 or G49) is specified ina block alone, the offset value is either added or cancelled realtime..**

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.

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

$$\text{Feedrate of Z axis} = \text{spindle speed} \times \text{thread lead}$$

**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 P294~297.

**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 in 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

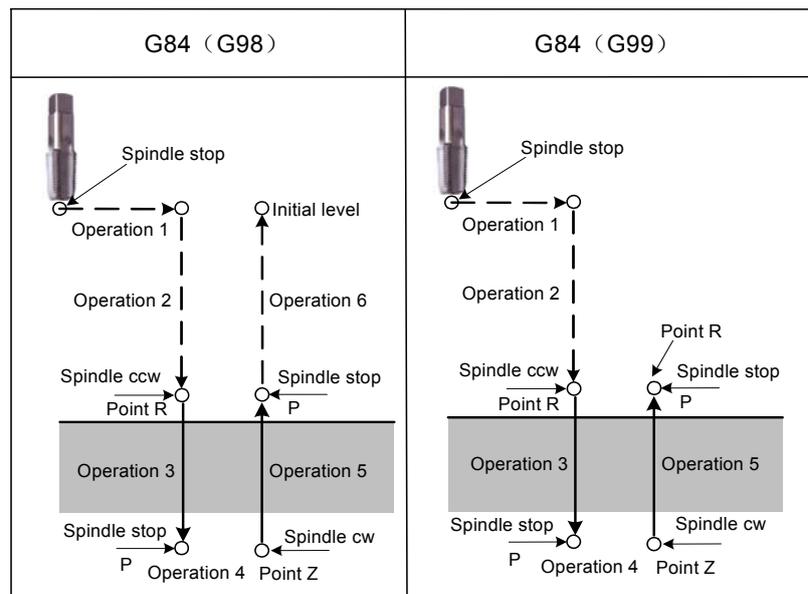
Z\_: In incremental programming it specifies the distance from point R level to the bottom of the hole; in absolute programming it specifies the absolute coordinate of the hole bottom.

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

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.

**Note 1 In current version, M00, M01, M02, M06, M30, M98, M99 will be executed after current statement is executed.**

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 (G43, G44 or G49) and canned cycle command are in a same block, the offset value is either added or cancelled while positioning to point R level; if if the tool length compensation (G43, G44 or G49) is specified ina block alone, the offset value is either added or cancelled realtime..**

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: feedrate/spindle speed.

$$\text{Feedrate of Z axis} = \text{spindle speed} \times \text{thread lead}$$

**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 |
| G84 Z-100 R-20 F1000; | Rigid tapping        |

**Restriction**

**F:** if the F value specified exceeds the upper limit of the cutting federate, the upper limit value is taken as final.

**S:** Alarm is issued if the rotation speed exceeds the max. speed of the gear specified which is set by number parameter P294~297.

**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 in 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 peck drilling rigid tapping, it performs multiple feeds till reaching 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.

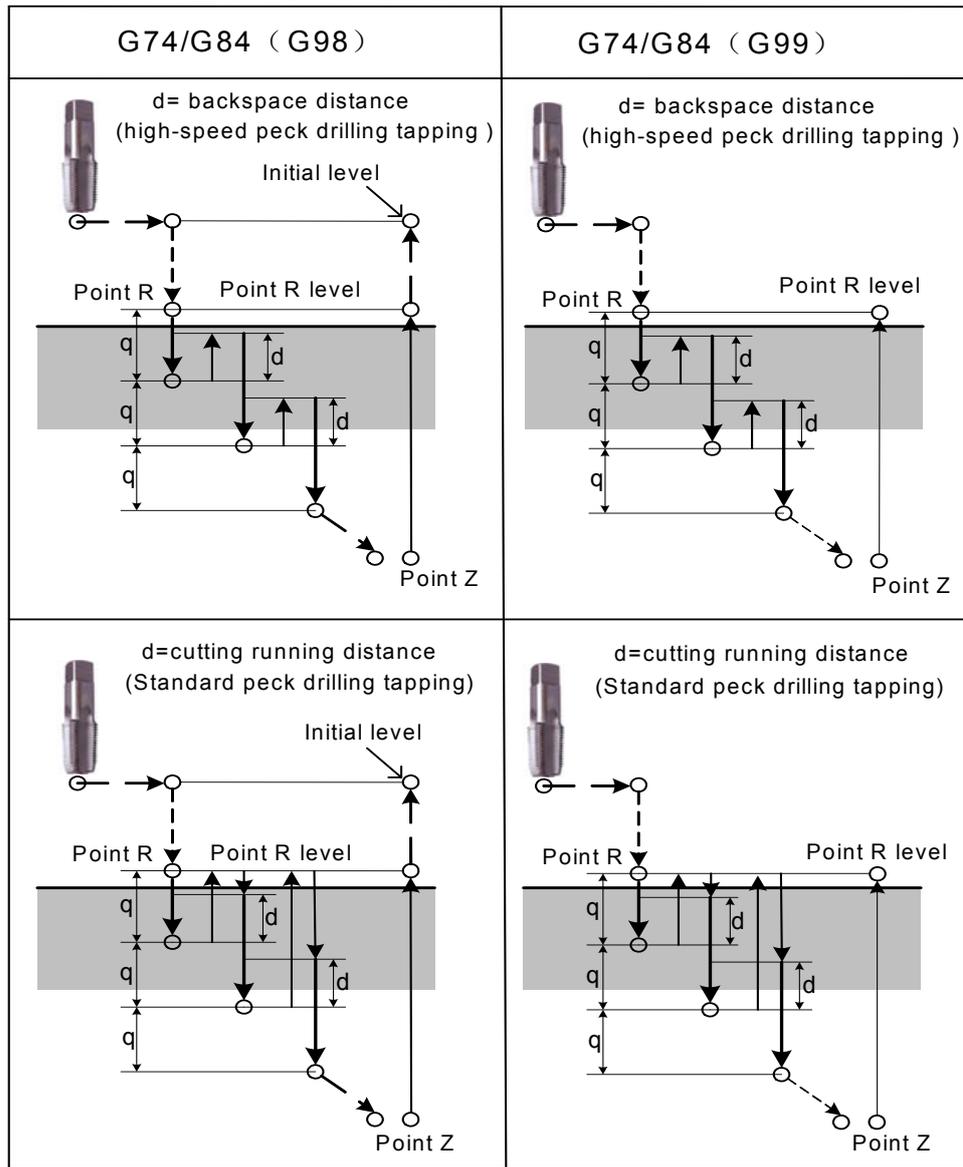
R\_: In incremental programming it specifies the distance from the initial level to point R level; in absolute programming it specifies the absolute coordinate of point R.

Q: Cutting depth for each cutting feed.

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

F\_: Cutting feedrate.

K\_: Number of repeats



**Fig. 4-4-21-1**

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 set by bit parameter NO: 44#5.

When bit parameter NO: 44#5 is 1, the mode is high-speed peck drilling tapping cycle: After positioning along X and Y axis, rapid 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 is retracted for a distance d (set by number parameter P284), the override in rigid tapping retraction is set by bit parameter NO:44#4, and the retraction rate override is set by bit parameter NO:45#3, a same time constant for rigid tapping feed and retraction is set by bit parameter NO:45#2, when it reaches the point Z ,the spindle stops, and then it rotates reversely for retraction.

When bit parameter NO;44#5 is 0, the mode is standard peck drilling tapping cycle: After positioning along X and Y axis, rapid traverse to point R level is performed. 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 retraction rate

override is set 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 retraction is set by bit parameter NO:45#2, when it reaches the Z point ,the spindle stops, and then it 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 in 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 in 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:**

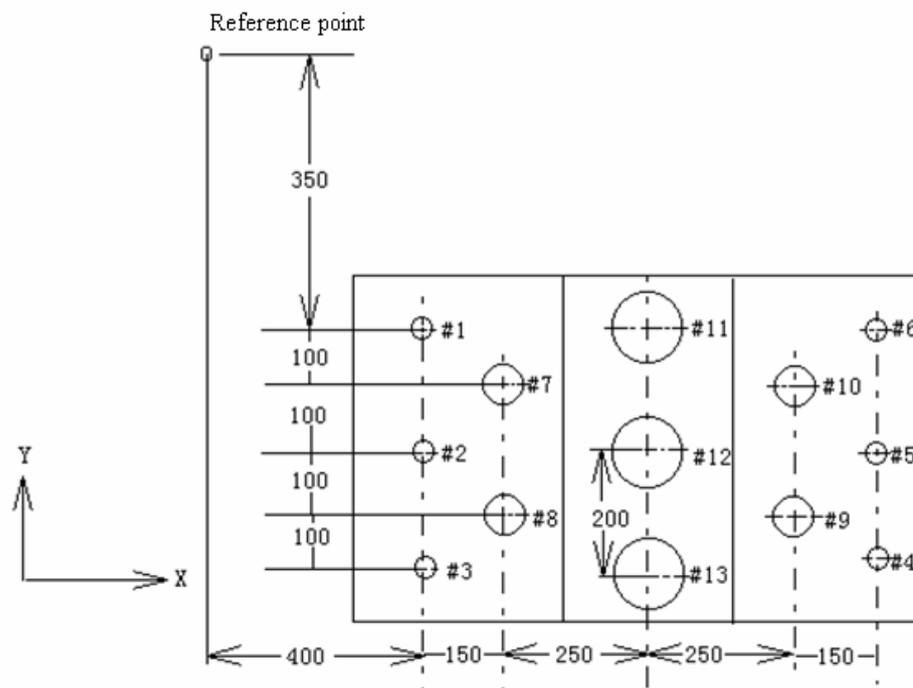
All canned cycles are cancelled for normal operation. Point R and point Z are cancelled too. Other drilling and boring data are also cancelled.

Example:

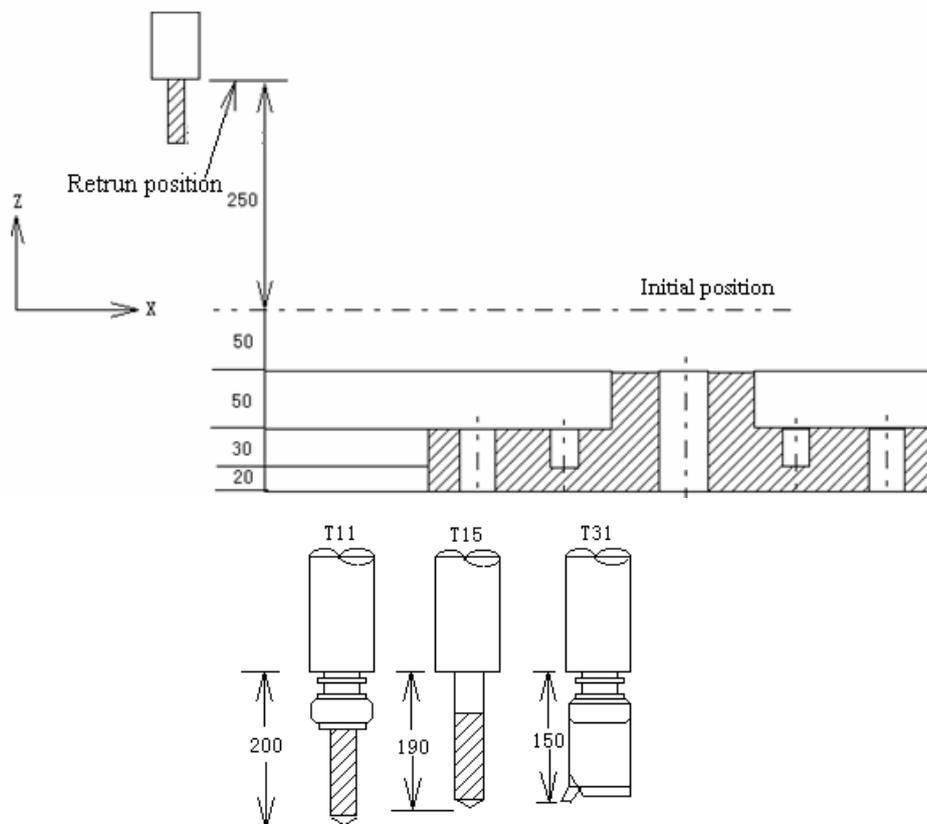
|  |  |
|--|--|
| M3 S100                                      | Spindle running start                                  |
| G90 G99 G88 X300. Y-250. Z-150. R-120. F120. |  |
|  | Positioning, bore hole 1, then return to point R       |
| Y-550.;                                      | Positioning, bore 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 of a  $\Phi 10$  hole
- # 7~10... drilling of a  $\Phi 20$  hole
- #11~13.. boring of a  $\Phi 95$  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 ;<br>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 ;<br>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 ;<br>Z-153 R47 F50 ;      | Positioning, then #11 drilling and point R level return     |
| N024 G91 Y-200 ;<br>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~H128 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:

Table 4-5-1-1

|                               | Range                      |
|-------------------------------|----------------------------|
| Offset value H ( mm input )   | -999.999 mm~+999.999mm     |
| Offset value H ( inch input ) | -999.9999 inc~+999.9999inc |

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

The tool length compensation is ineffective before Z instruction.

**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

**Programming**

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

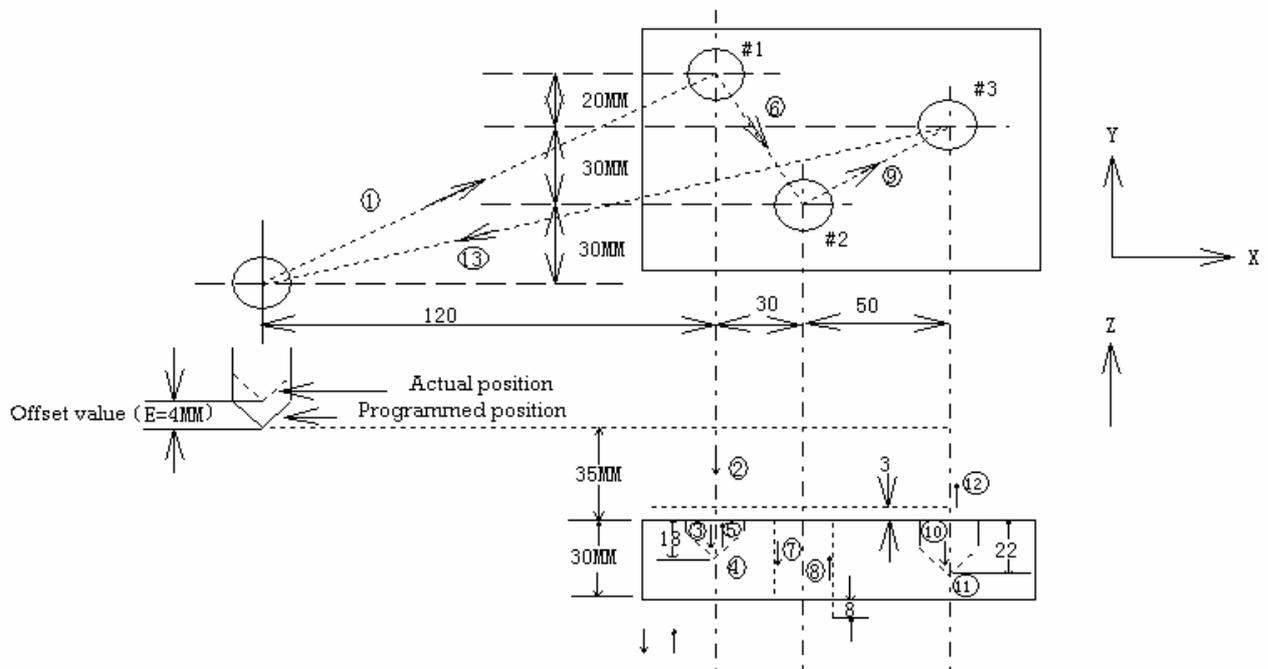
**Note 1: After B mode of tool length offset is executed along two or more axes, all the axes offset can be cancelled by G49, while only the axis offset perpendicular to a specified plane can be cancelled by H00.**

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 reference point. In tool length offset, the offset vector cancelled 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 ;

**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.

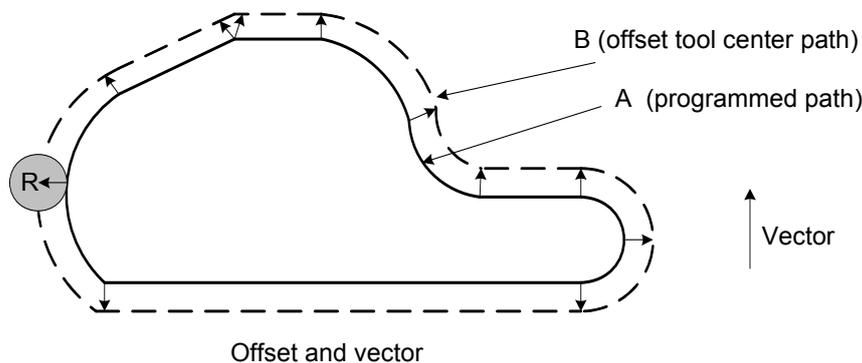
G42 specifies the right offset of the tool moving.

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)**

**Programming**

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~D127 as required. The diameter or radius value of it can be set by bit parameter No.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.

Offset value corresponding to the offset number can be presetted in offset memory with LCD/MDI.

The range of the offset value is as follows:

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

|                             | Range                    |
|-----------------------------|--------------------------|
| Offset value D (mm input)   | -999.999mm~999.999mm     |
| Offset value D (inch input) | -999.9999inc~999.9999inc |

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

Alterration for compensation plane can be executed after canceling the offset mode, or alarm occurs.

**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 combination 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, G28or G30 command in tool radius offset mode**

If G53, G28or G30 command is specified in tool radius offset, the offset vector of tool radius offset axis is cancelled after it moves to the specified position (it is cancelled when moves to the position specified by G53, and cancelled at the reference point specified by G28, G30), but the modal code display is not switched to G40, the axes except the tool radius offset axis are not cancelled. If G53 and G41/G42 share a same block, all the axis radius offsets are cancelled after the axis moves to the specified position; if G28 or G30 shares a same block with G40, they are cancelled after axis moves to the reference point; In tool radius offset, the offset vector cancelled by G53, G28 or G30 will be restored in the next block in the buffer.

Note: in offset mode, G28, G30 can be set by bit parameter NO: 40.2 for offset temporary cancellation)

**Tool radius compensation cancel (G40)**

Use the following instruction to perform the linear motion from the old vector of the start point to the end point in G00, G01 mode:

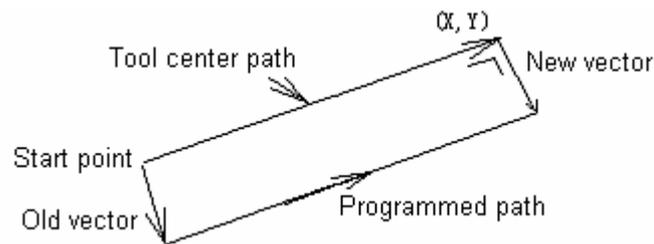
It performs linear movement from the old vector of start point to the end point. In G00 mode, the axes rapidly traverse to the end point. By using this instruction, the system enters into tool radius compensation cancel mode from tool radius compensation mode

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 instuction 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.....;

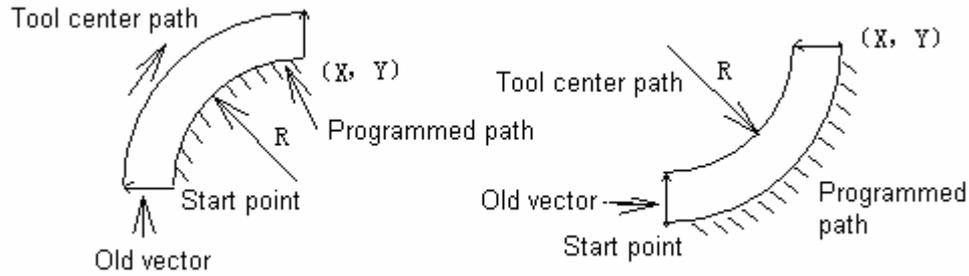
.....

.....

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 (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**

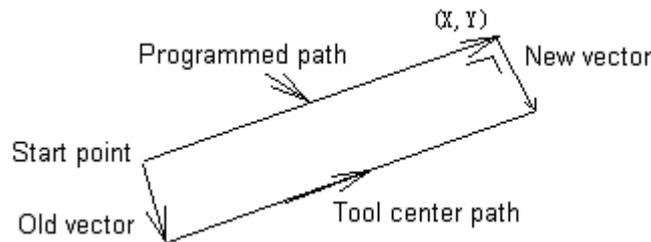
**Tool radius compensation right (G42)**

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**

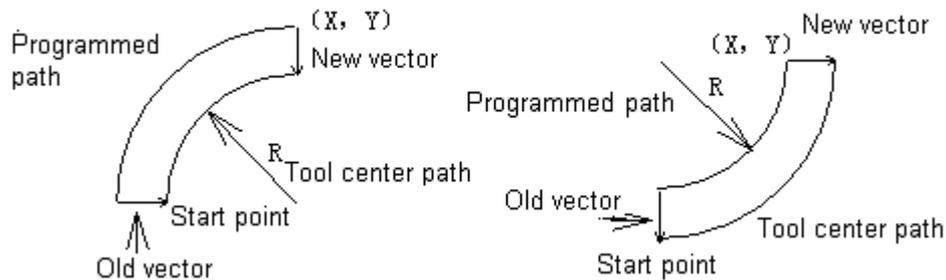


Fig. 4-5-2-5

**6、Precautions on offset**

**(A) Specification of offset number**

G41, G42 and G40 are modal codes. 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:

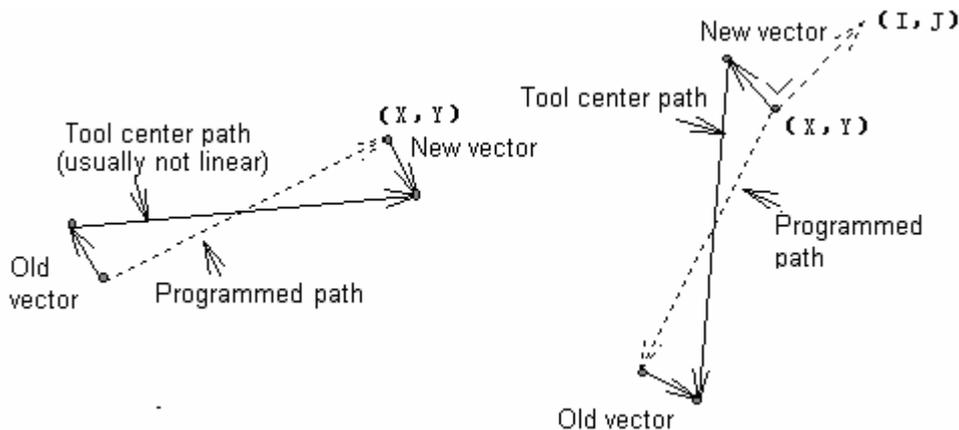


Fig. 4-5-2-6

G1G41 D\_\_X\_\_Y\_\_;

G42 D\_\_X\_\_Y\_\_;

.....

.....

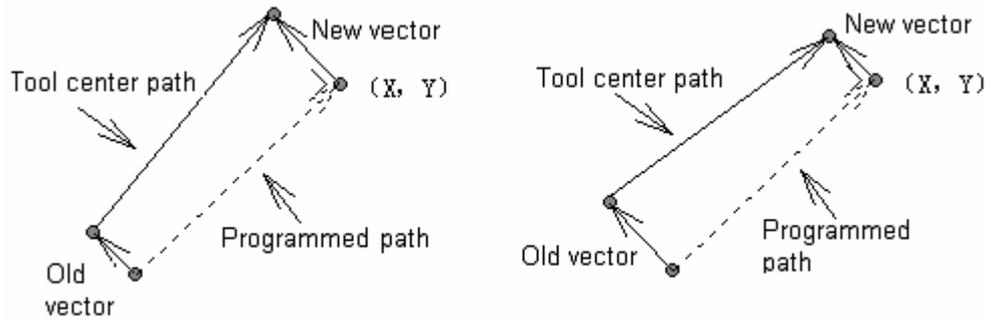
G1G42 D\_\_X\_\_Y\_\_;

G41 D\_\_X\_\_Y\_\_;

**(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:



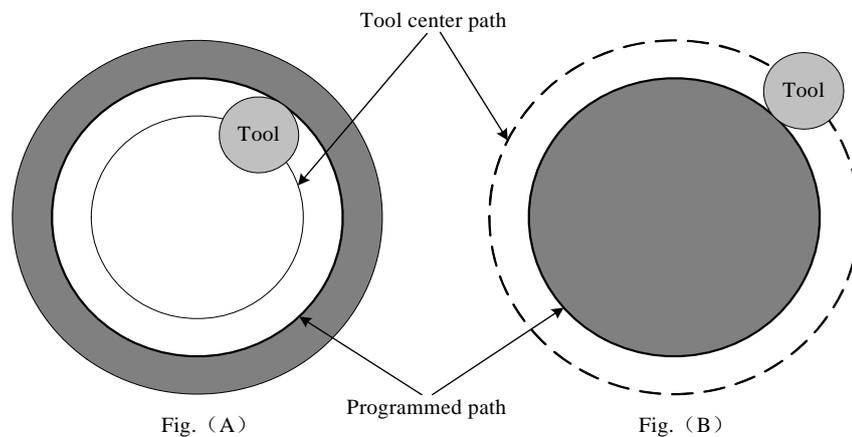
**Fig. 4-5-2-7 The change of offset value**

**(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 for 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 ;

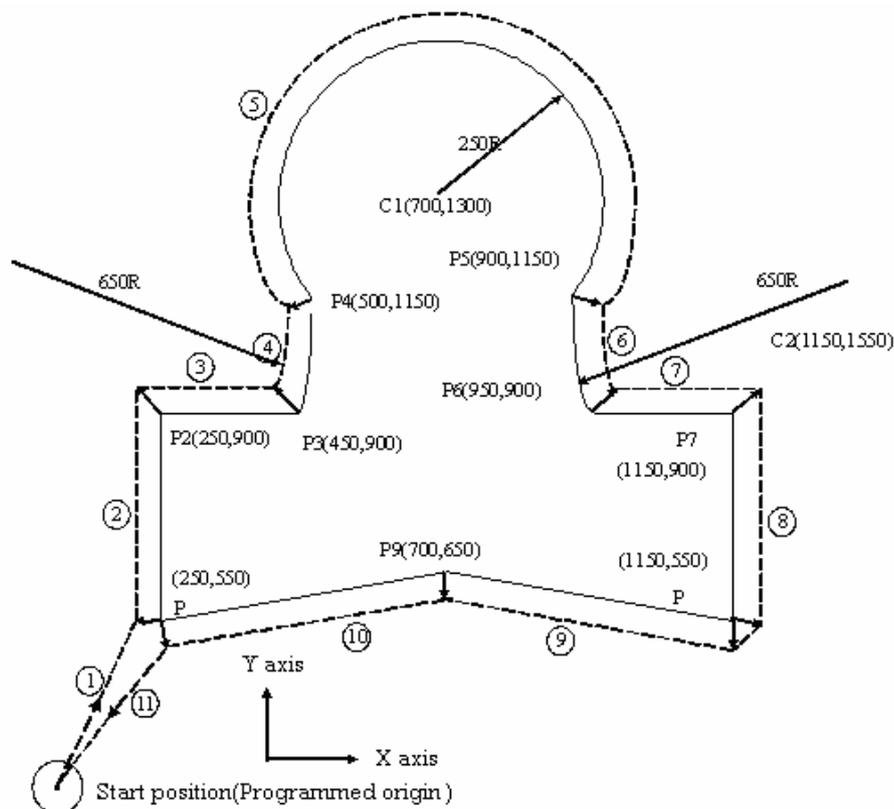


Fig. 4-5-2-9

### 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°, it is called inner side, when the angle is between 0° and 180°, it is called outer side.

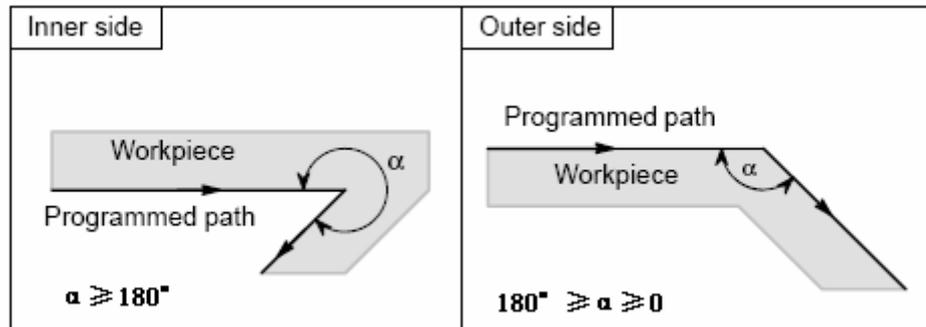


Fig. 4-5-3-1

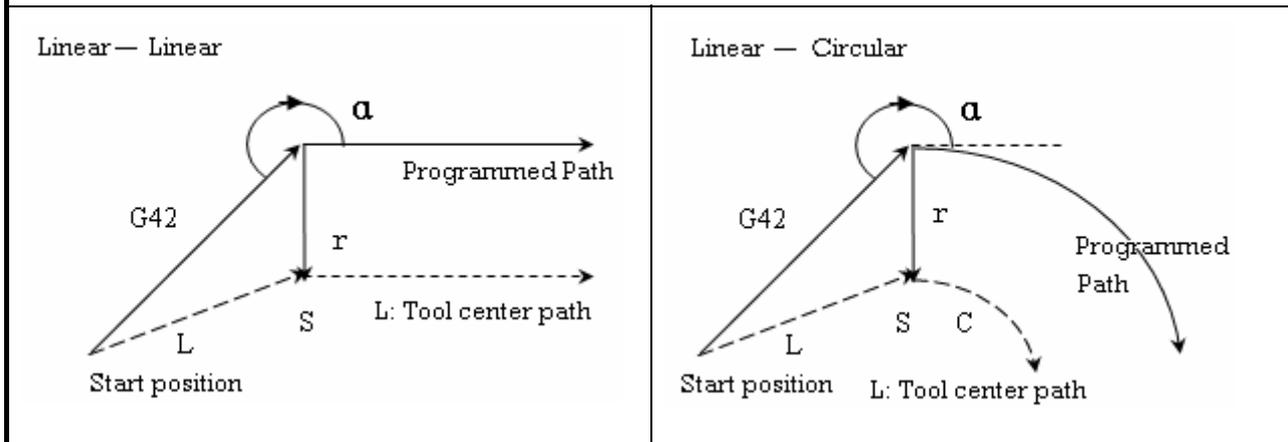
Meanings of symbols:

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

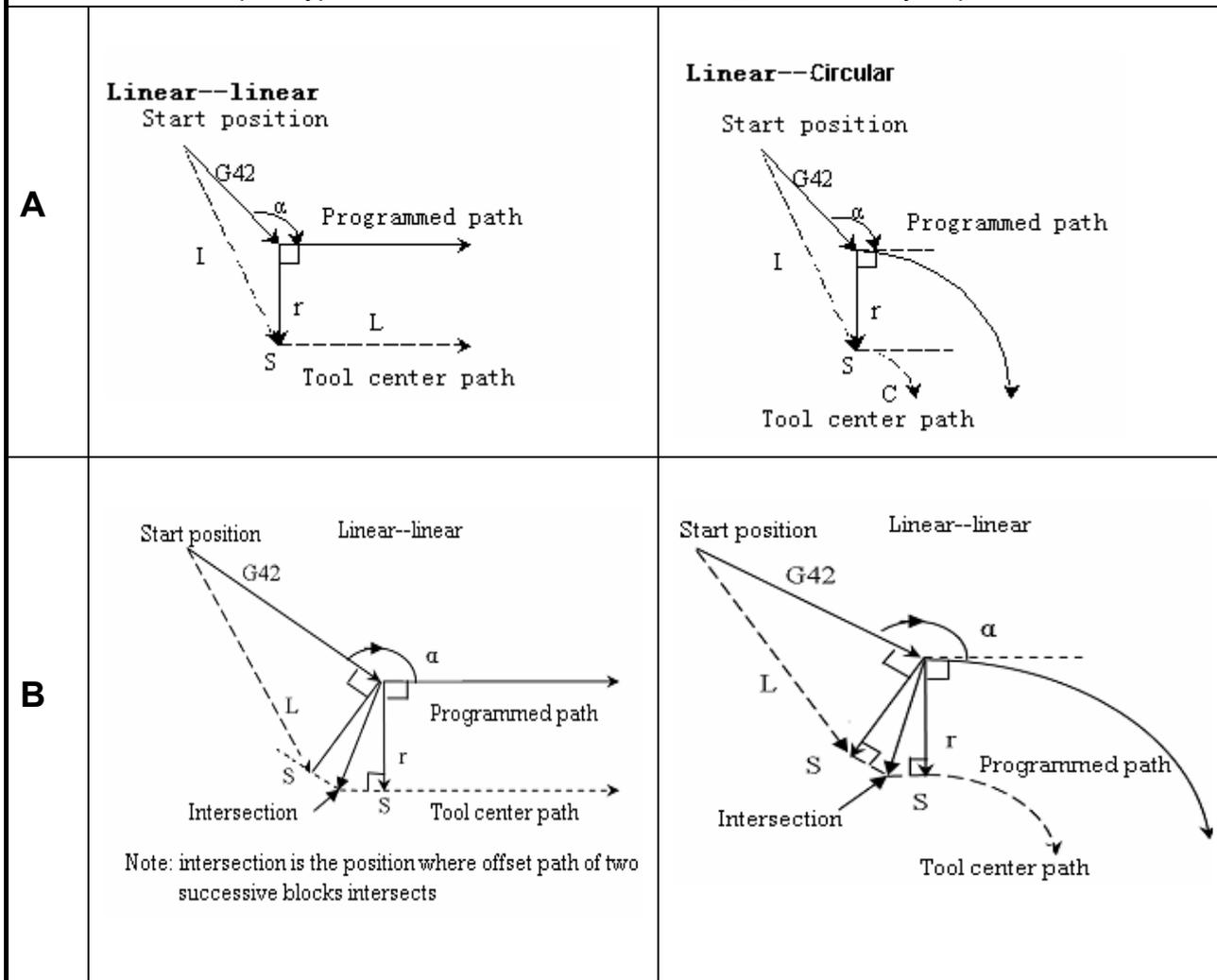
1. Tool movement in start-up When the offset cancel mode is changed to offset mode, the tool moves as illustrated below(start-up):

**(a) Tool movement around an inner side of a corner ( $\alpha \geq 180^\circ$ )**



**(b) Tool movement around an outer side of a corner at an obtuse angle ( $180^\circ > \alpha \geq 90^\circ$ )**

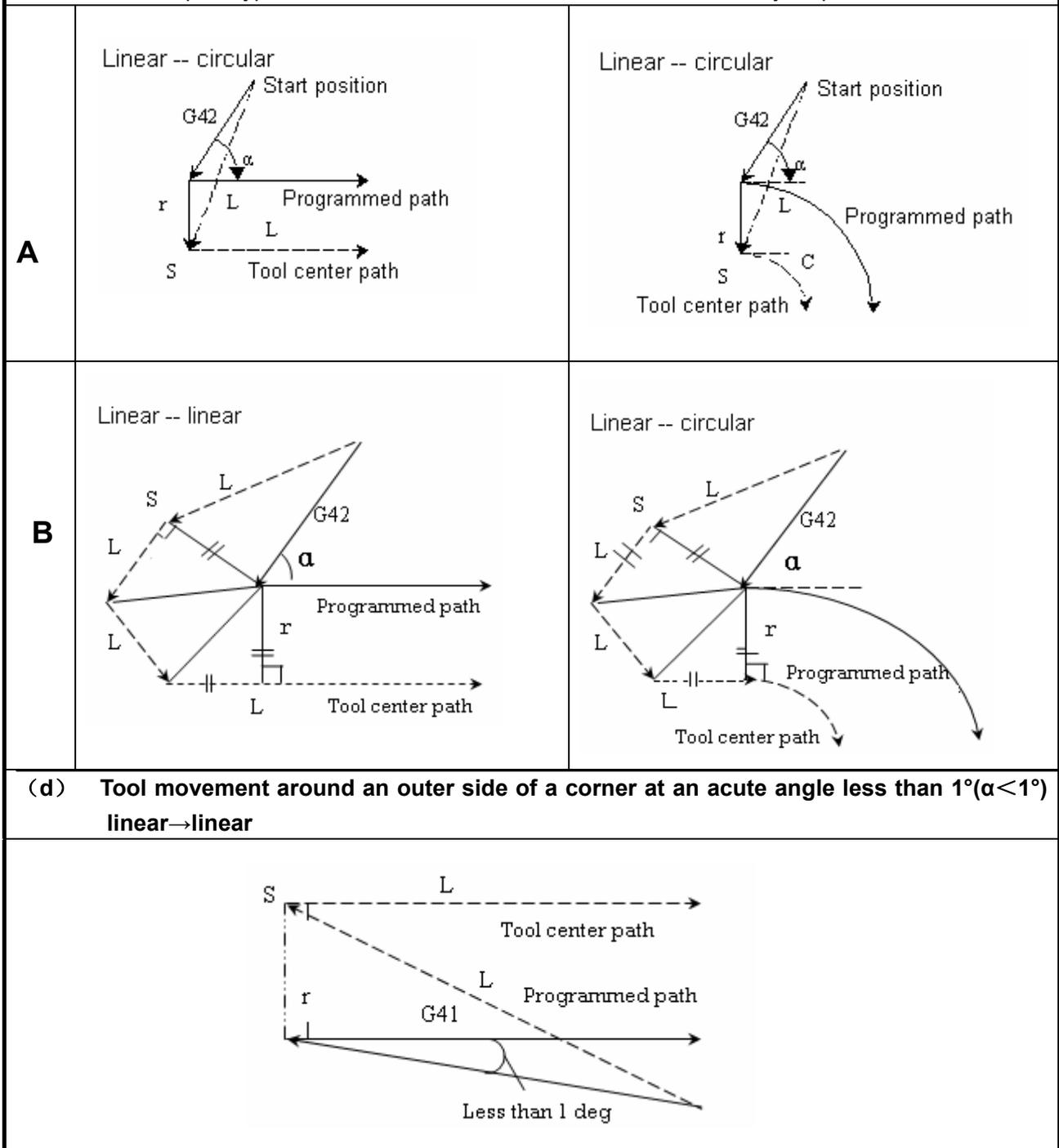
: There are 2 tool path types at offset start or cancel: A and B, which is set by bit parameter No: 40#0.



**Programming**

**(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.



**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:

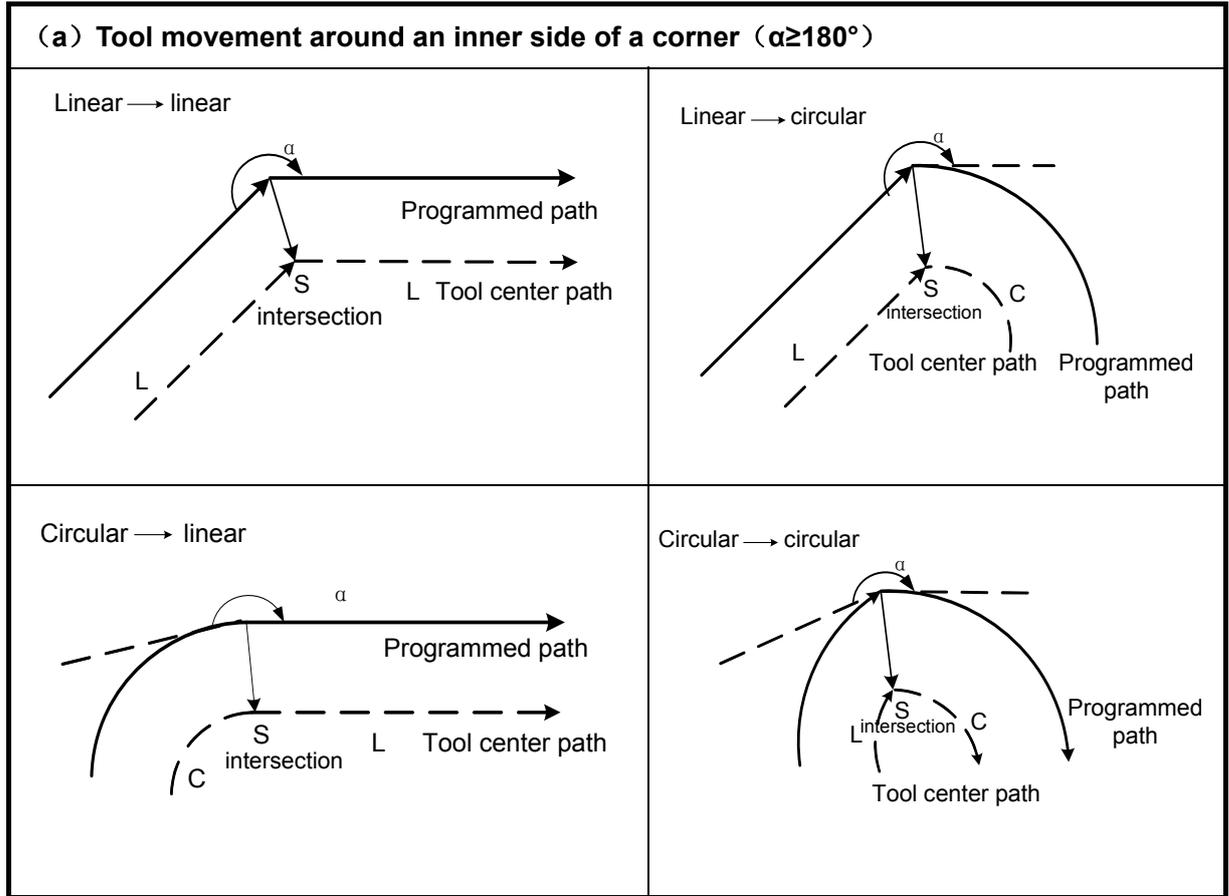


Fig. 4-5-3-3

3. Special case:

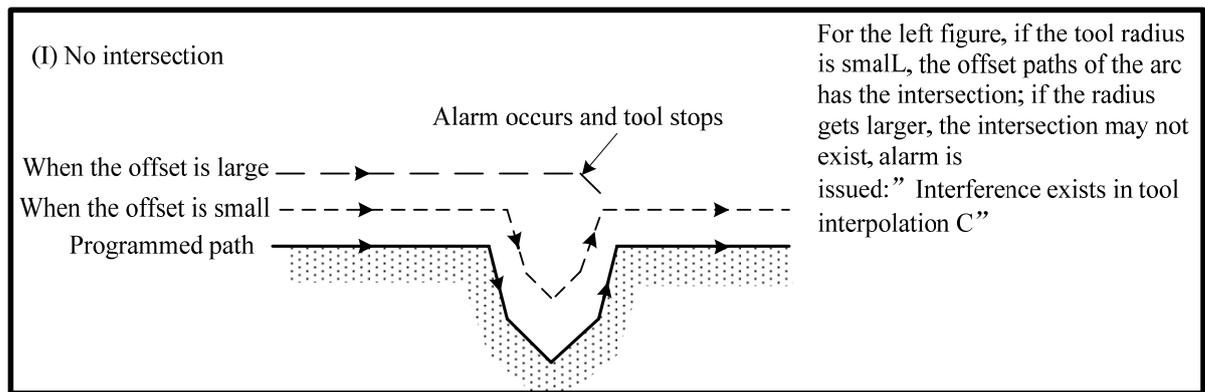


Fig. 4-5-3-4

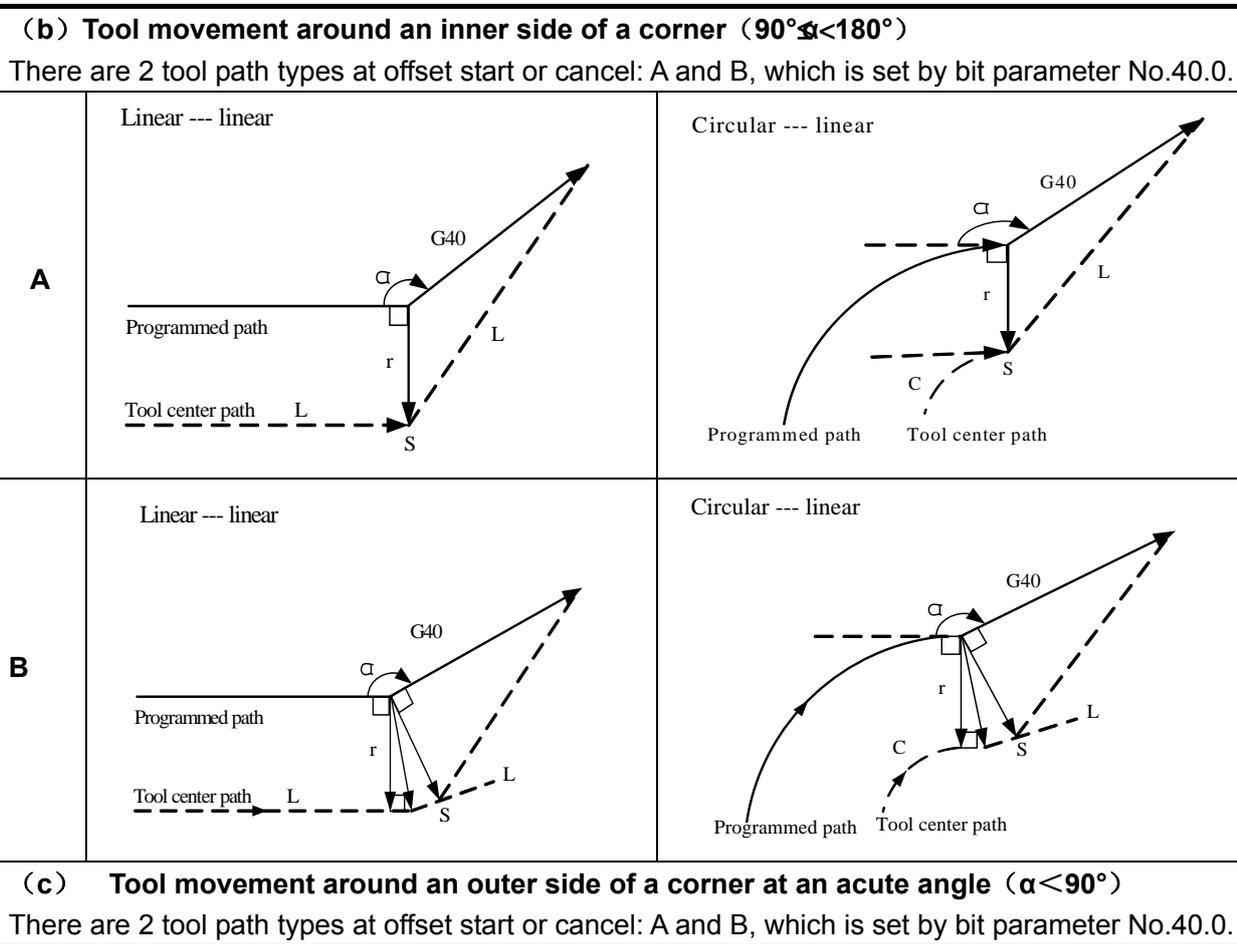
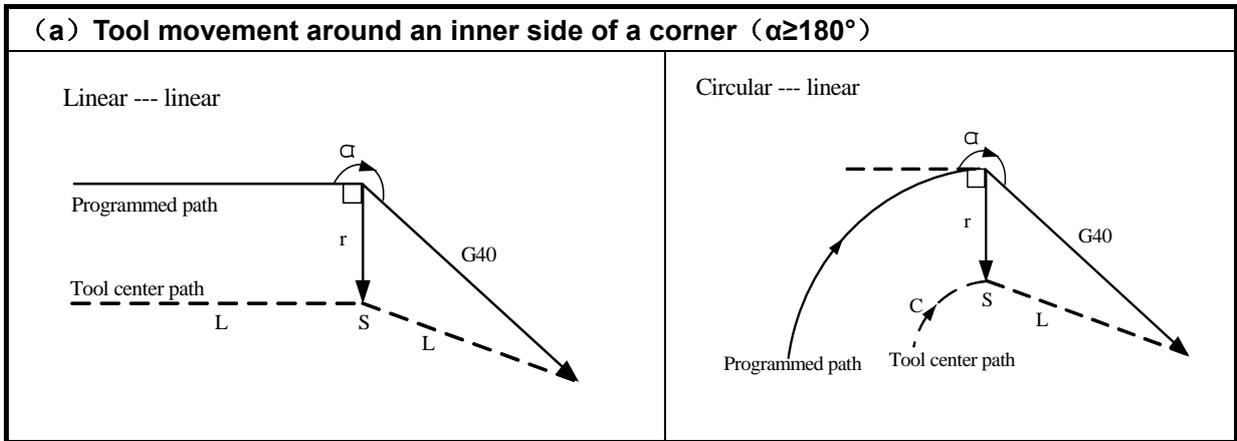
4. Tool movement in offset cancel mode

In offset mode, when the block complies to any of the following condition is executed, the system enters offset cancell 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



**(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.

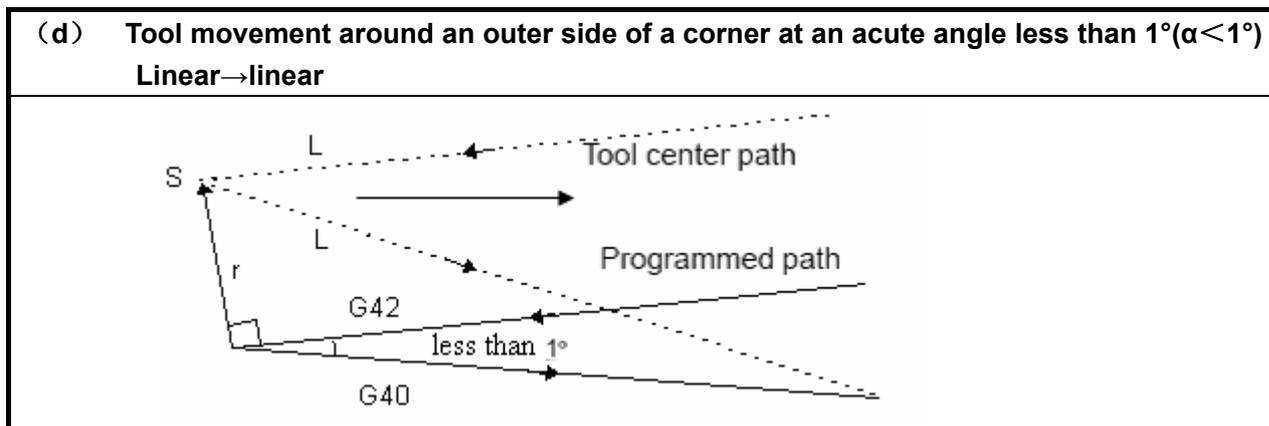
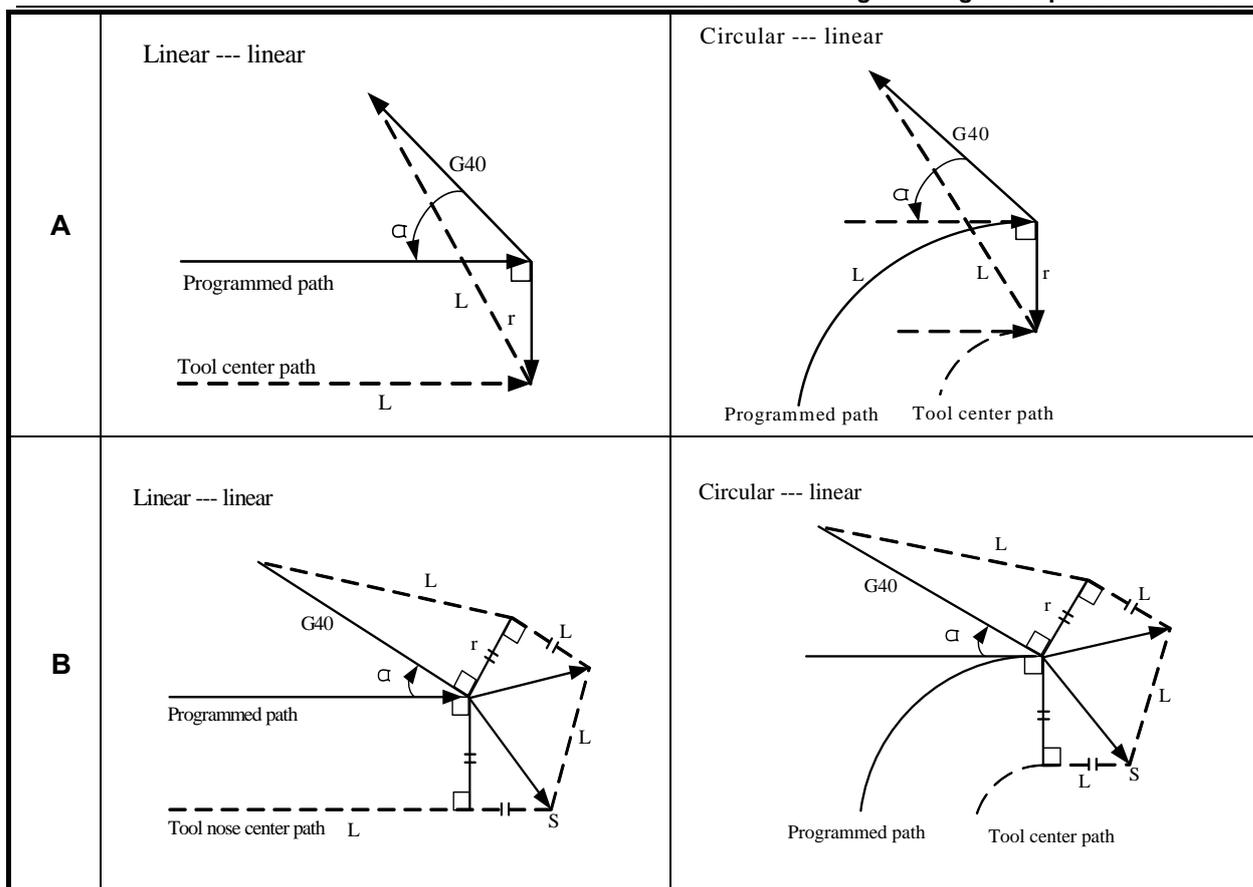


Fig. 4-5-3-5

**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:

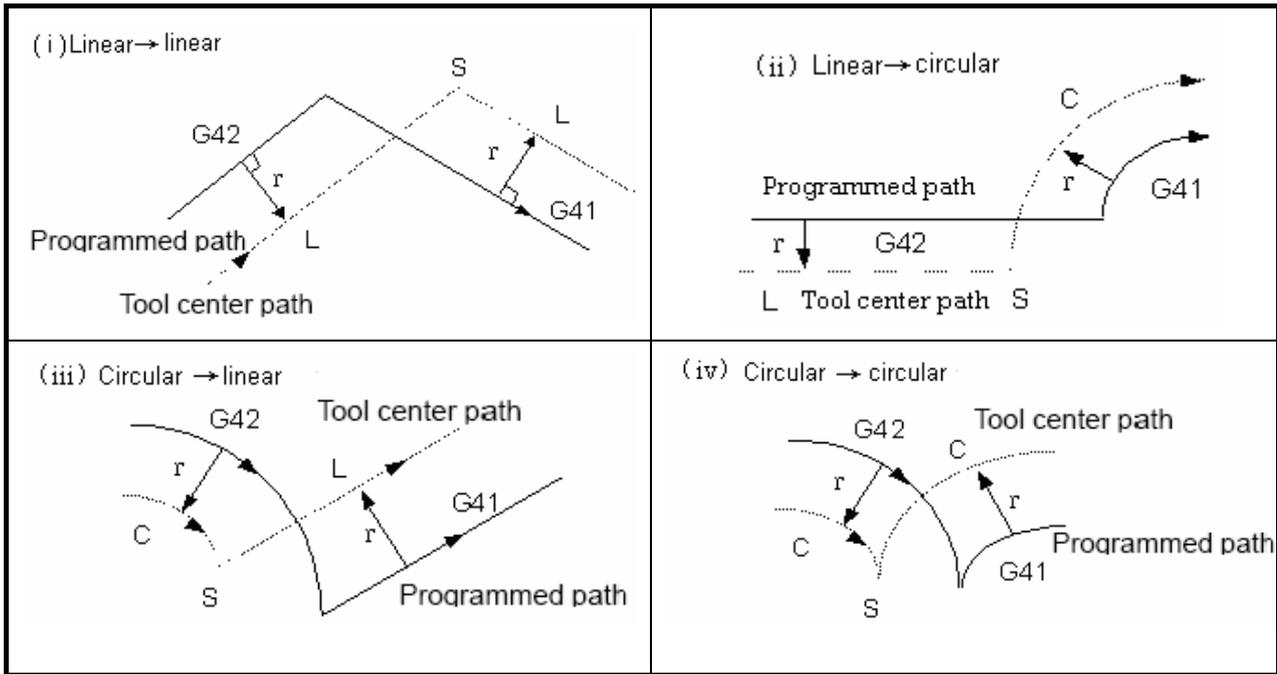
**Table 4-5-3-1**

| Sign of offset value<br>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

**Programming**

when the offset direction is changed. The following offset value is assumed to be positive.

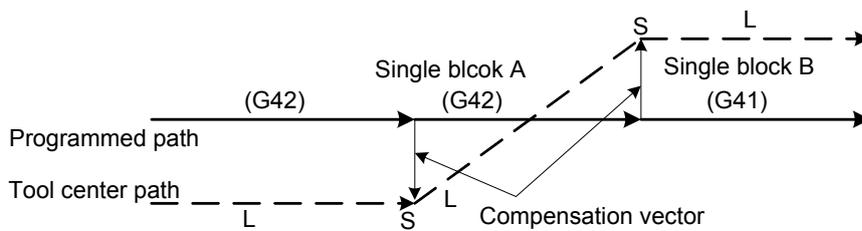


**Fig. 4-5-3-6**

(v) When the tool compensation is executed normally without an intersection

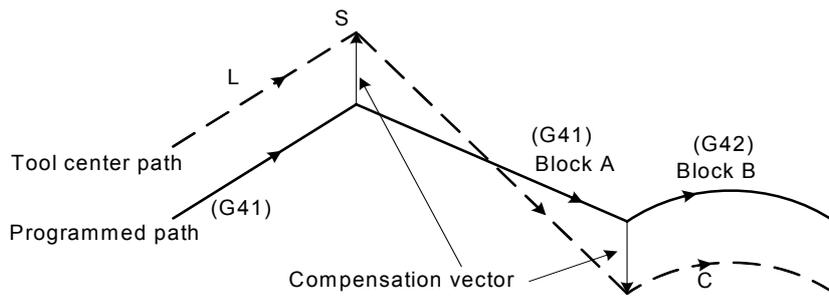
When changing the offset direction from block A to block B using G41 and G42, if intersection of the offset path is not required, the vector normal to block B is created at the start point.

**(1) Linear---- linear**



**Fig. 4-5-3-7**

**(2) Linear----- circular**



**Fig. 4-5-3-8**

(3) Circular----- circular

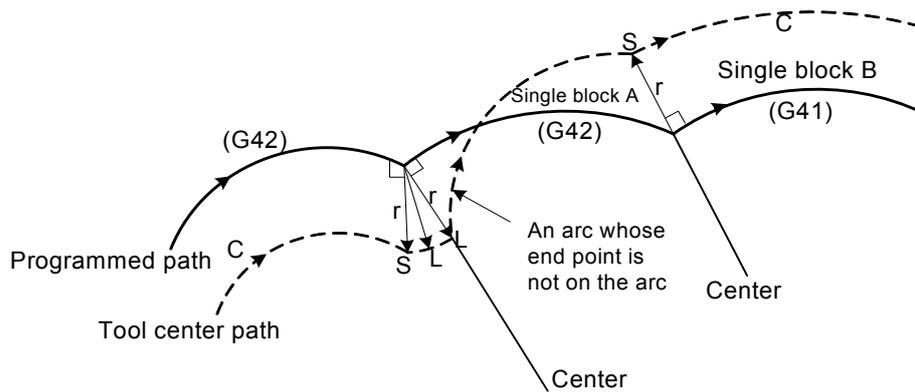
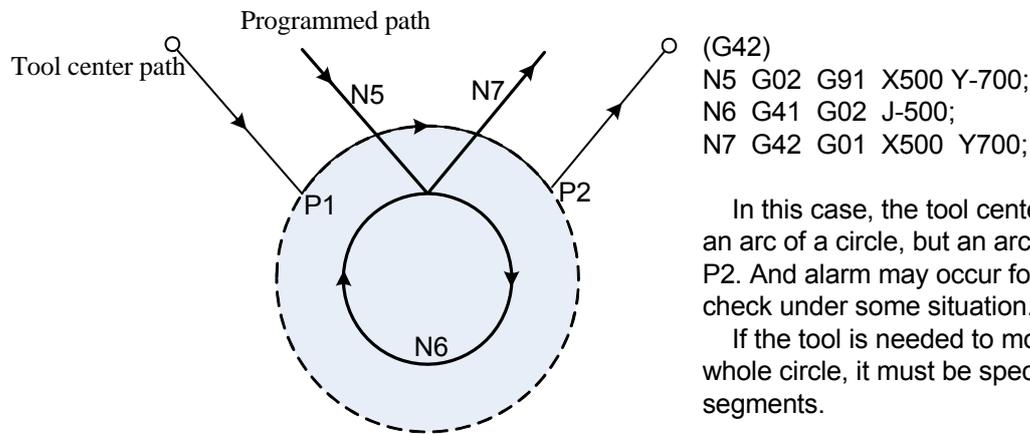


Fig. 4-5-3-9

(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.



In this case, the tool center path is not an arc of a circle, but an arc from P1 to P2. And alarm may occur for interference check under some situation.

If the tool is needed to move by the whole circle, it must be specified by segments.

Fig. 4-5-3-10

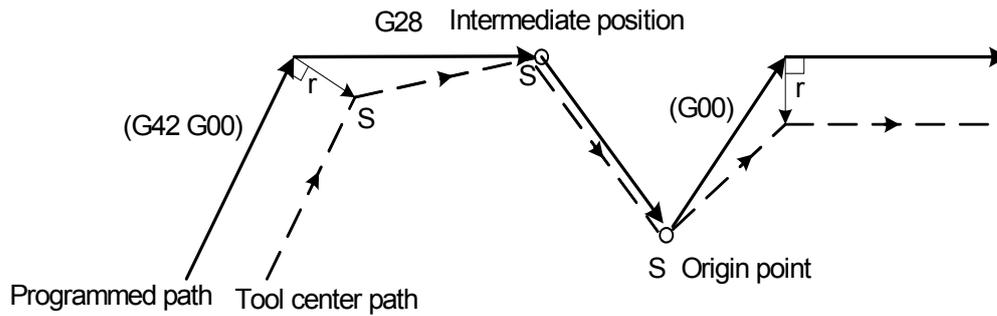
6. Temporary offset cancel

In offset mode, whether the offset is temporarily cancelled or not in intermediate point is set by parameter No:40#2 when G28, G30 is specified.

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.

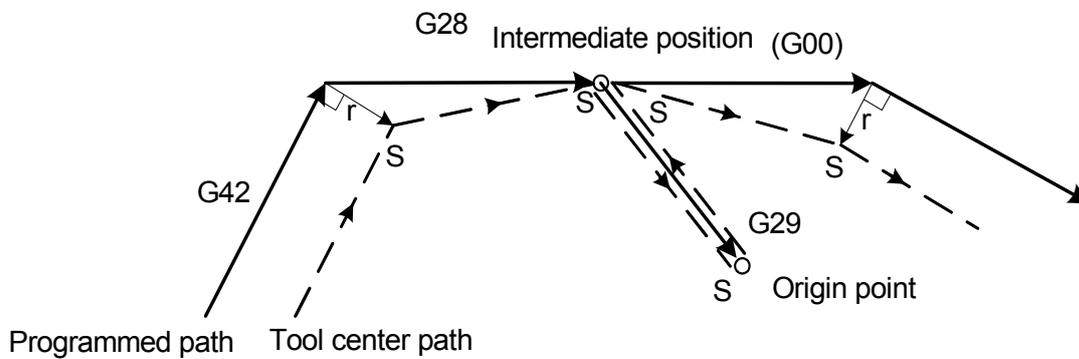


**Fig. 4-5-3-11**

b ) G29 automatic return from origin point

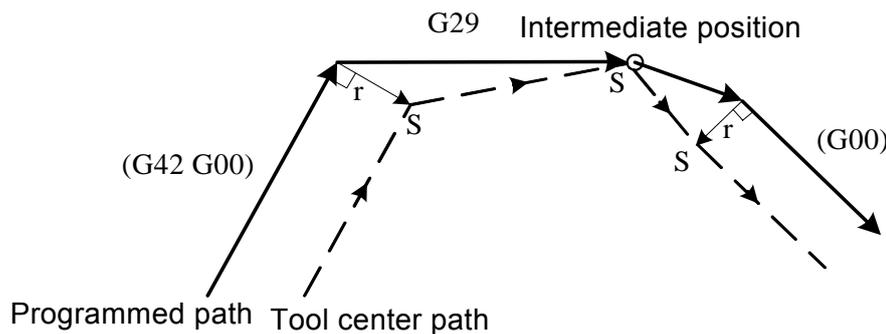
If G29 is specified in offset mode, the offset is cancelled at the intermediate position and automatically restored at the next block.

If it is specified immediately after G28:



**Fig. 4-5-3-12**

If it is not specified immediately after G28:



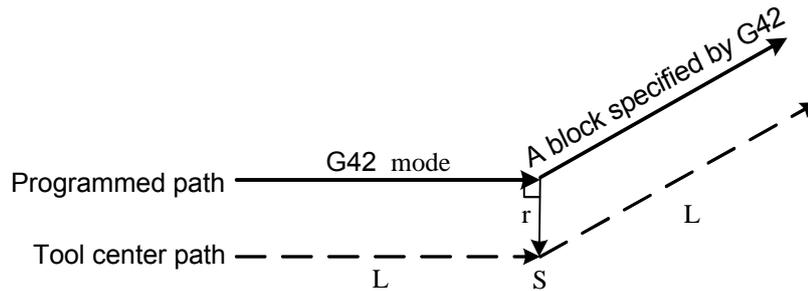
**Fig. 4-5-3-13**

**7. Tool radius compensation G code in offset mode**

In offset mode, if the tool radius compensation G code (G41, G42) is specified, 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.

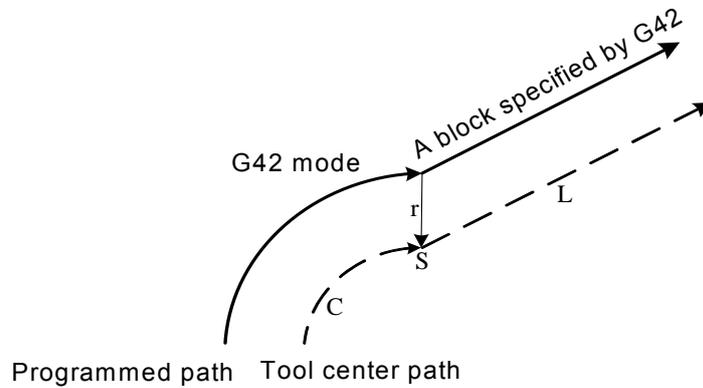
If this code is specified in a circular instruction, correct motion will not be obtained. Refer to ( 5 ) for offset direction change by tool radius compensation G ( G41, G42 )

Linear----- linear



**Fig. 4-5-3-14**

Circular----- linear

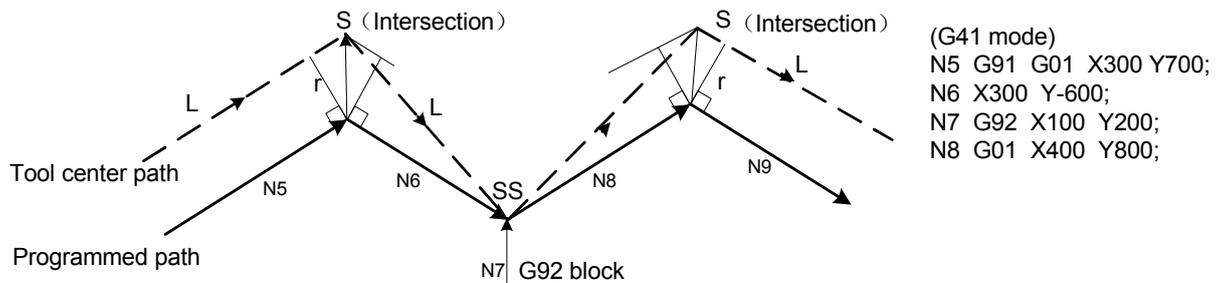


**Fig. 4-5-3-15**

**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**

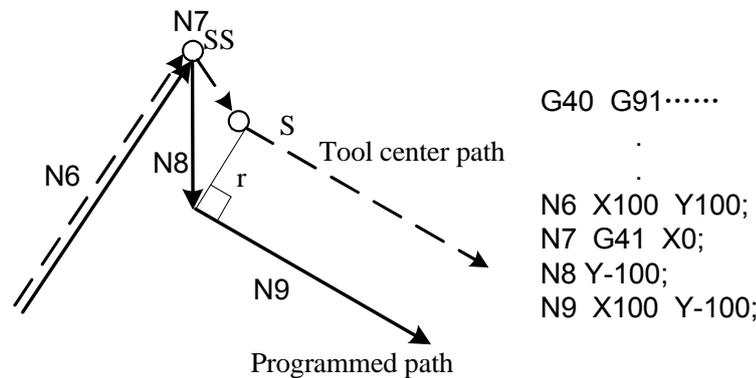
The following blocks have no tool movement. In these blocks, the tool will not move even if tool

radius compensation is effective.

- (1) M05 ; ..... M code output
- (2) S21 ; ..... S code output
- (3) G04 X10; ..... 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**

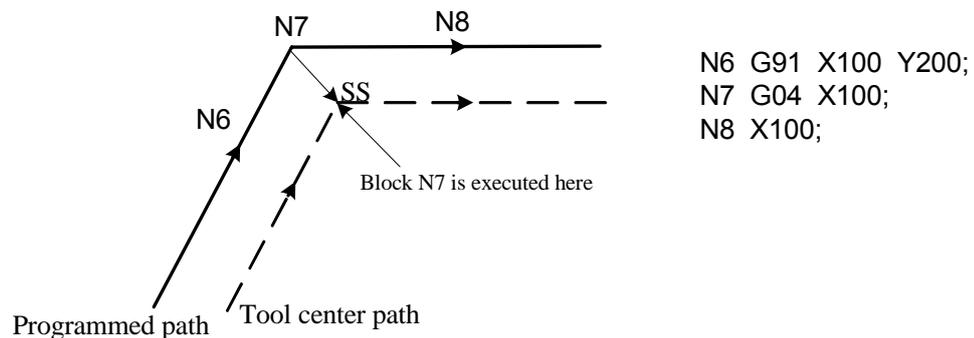
If the tool movement is not made by the start-up block, it will be done by the next moving instruction block by the system.



**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 of 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.

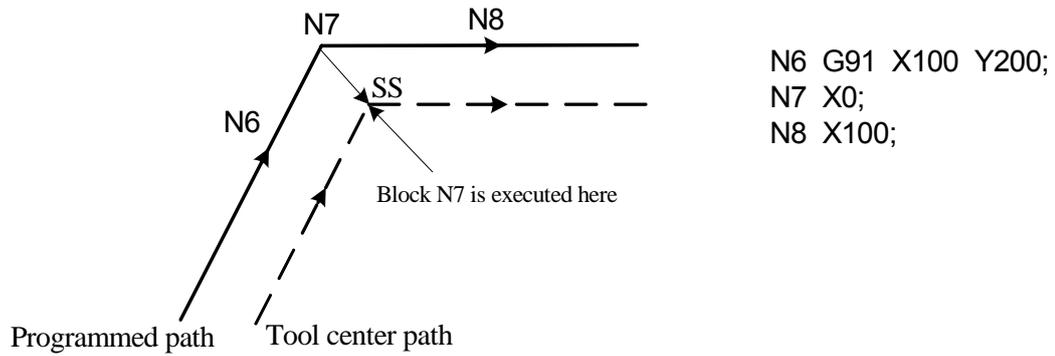


Fig. 4-5-3-19

**Note** The blocks above are executed in G1, G42 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.

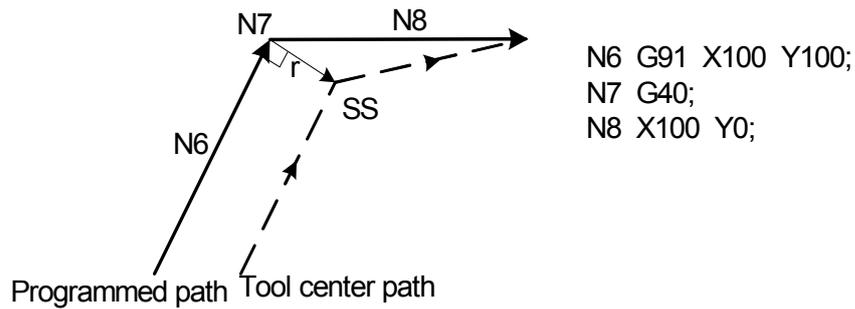


Fig. 4-5-3-20

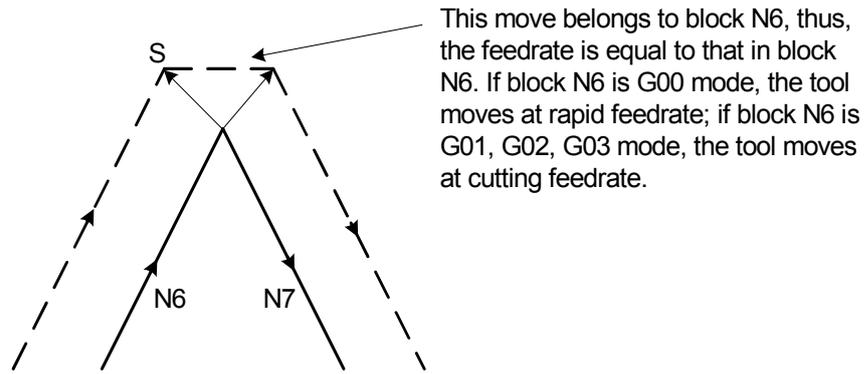
### 10. Corner movement

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.

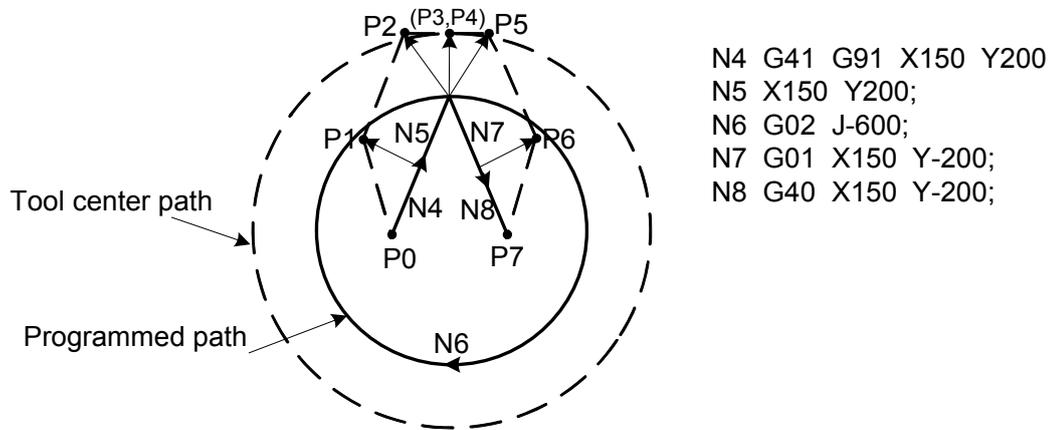
If these vectors are not consistent, a movement around the corner is generated, which belongs to the front block.

**Programming**



**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 → 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.41.3.

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 ①

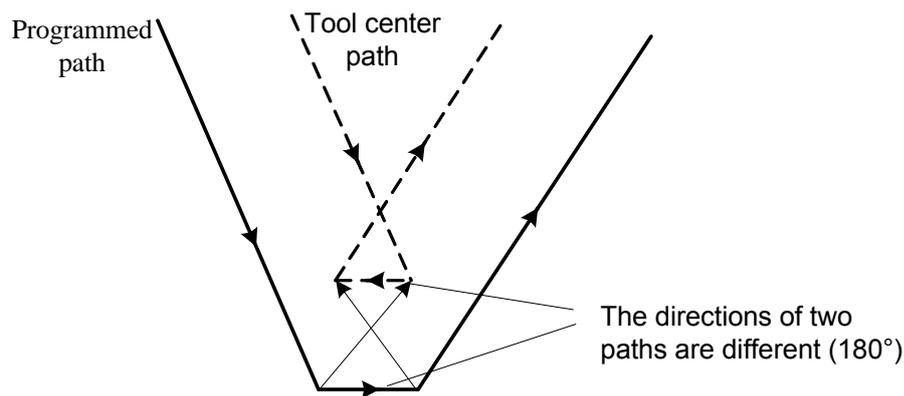


Fig. 4-5-3-23

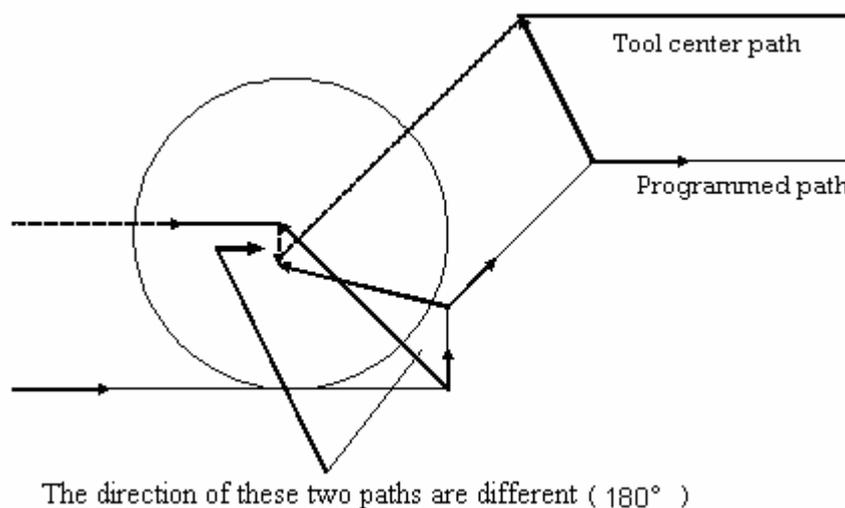


Fig. 4-5-3-24

## 12. Manual operation

See the manual operation in Operation section for the manual tool radius compensation.

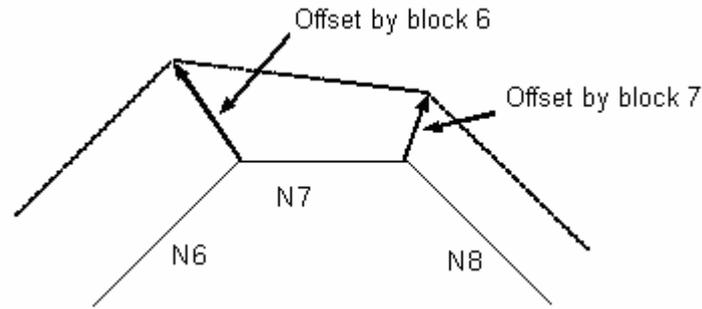
## 13. Precautions for offset

(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.

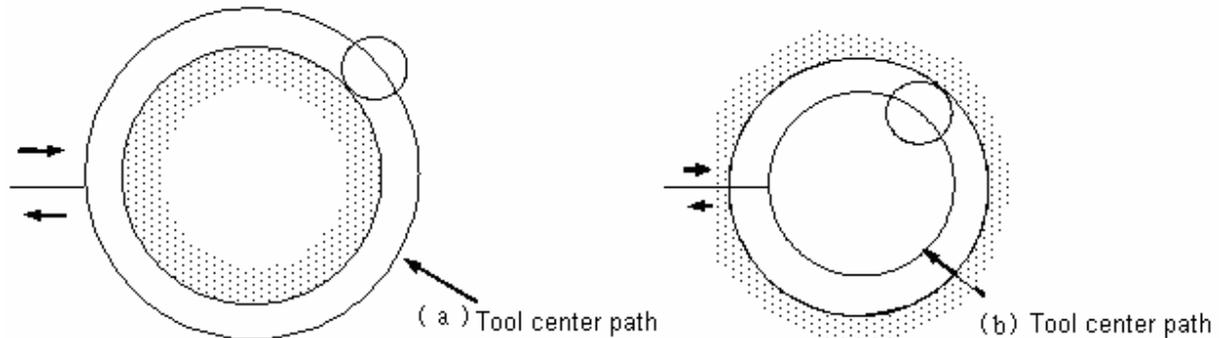


**Fig. 4-5-3-25**

(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.

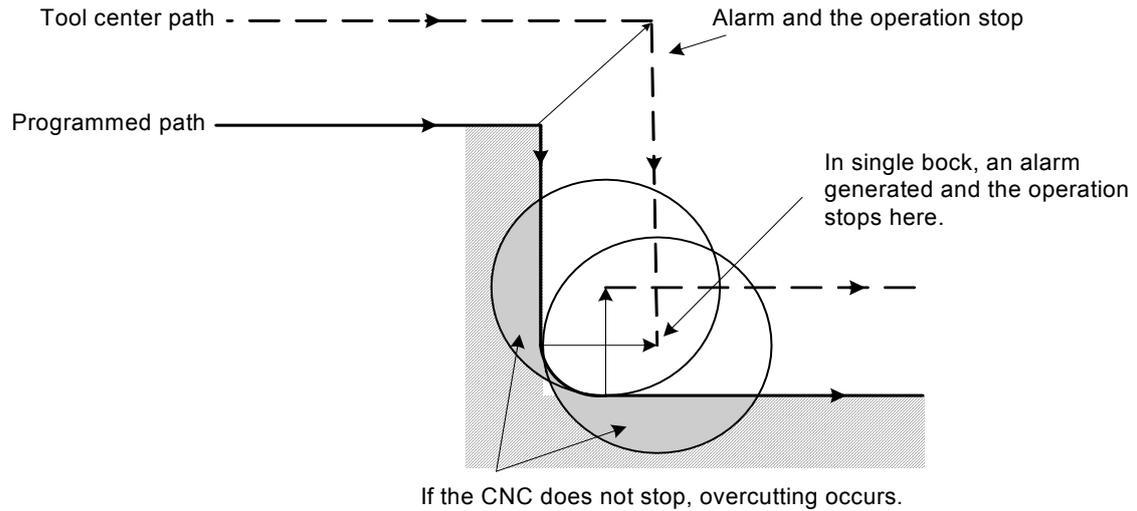


Fig. 4-5-3-27

(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.

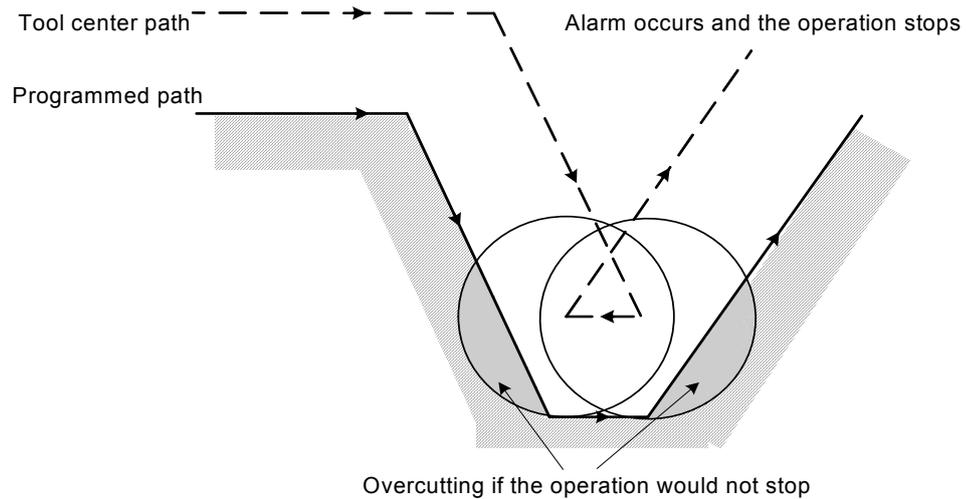
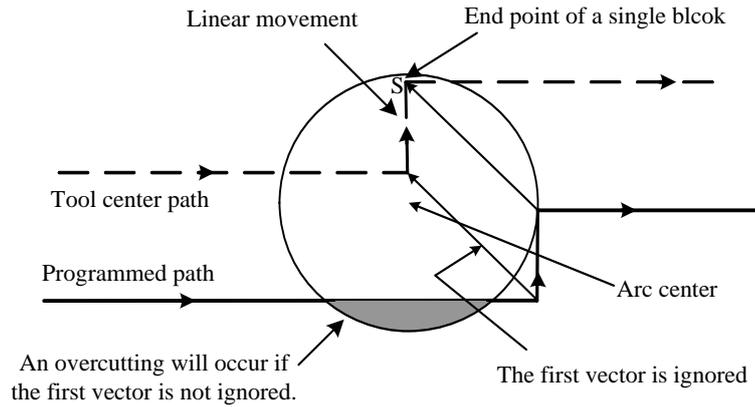


Fig. 4-5-3-28

(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**

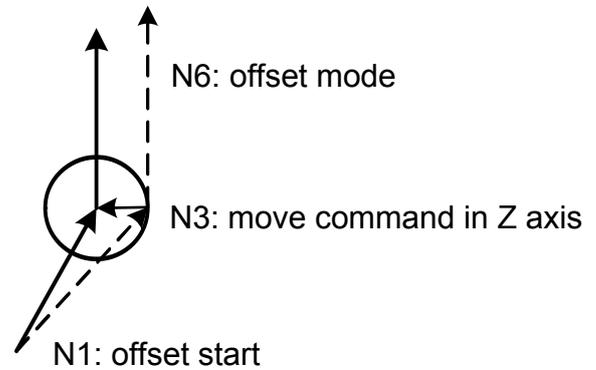
It is usually used such a method that the tool is moved along the Z axis after the tool radius compensation is effected at some distance from the workpiece at the start of the machining. In the case above, if it is desired to divide the motion along the Z axis into rapid feed and cutting feed, follow 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 is entered 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)**

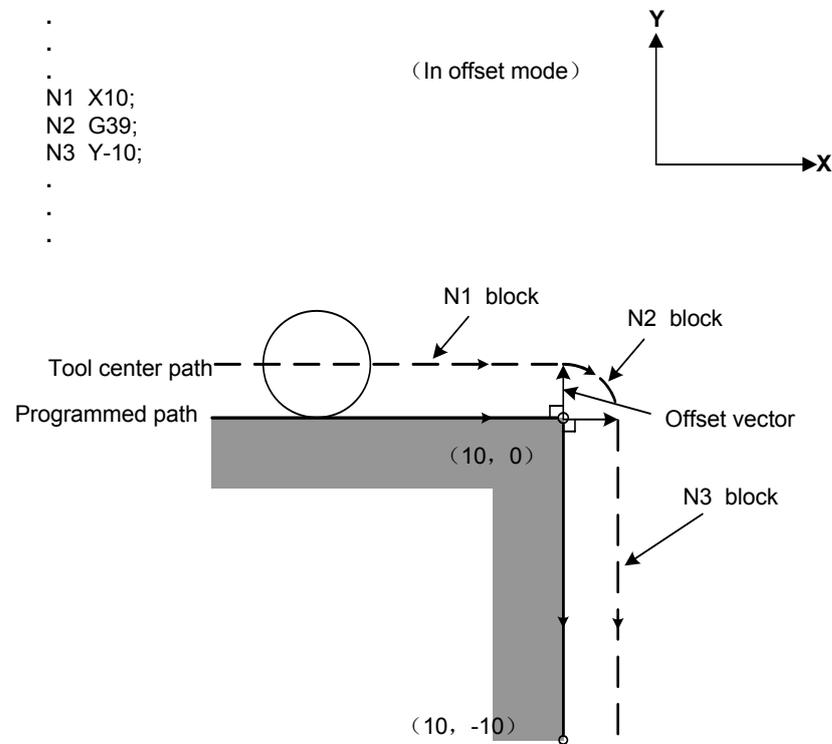
**Format:** G39 or  
           I\_ J\_  
           G39 I\_ K\_  
               J\_ K\_

**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.41.6.

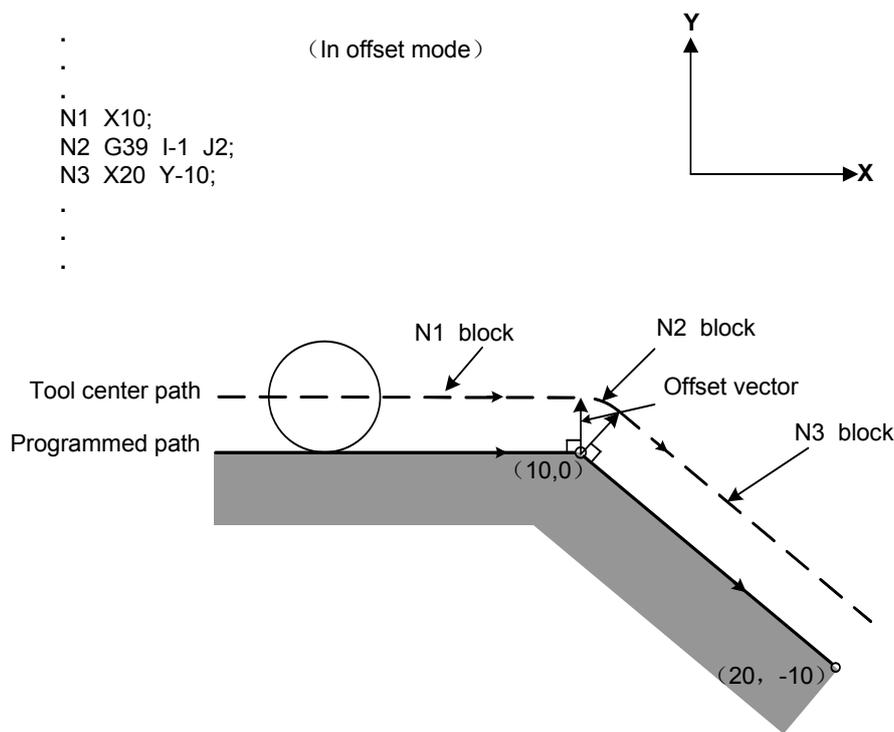
**Explanation:**

- 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 without I, J, K**

- 4、 When G39 is specified with I, J, K, the arc at the corner is formed so that the vector at the end of the arc is perpendicular to the vector defined by the I, J, K values. It is shown as follows:



**Fig. 4-5-4-2 G39 containing I, J, K**

#### 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 ±999.999mm; inch input ±99.9999 inch
- Wear offset: metric input ±99.999mm; inch input ±9.9999 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:

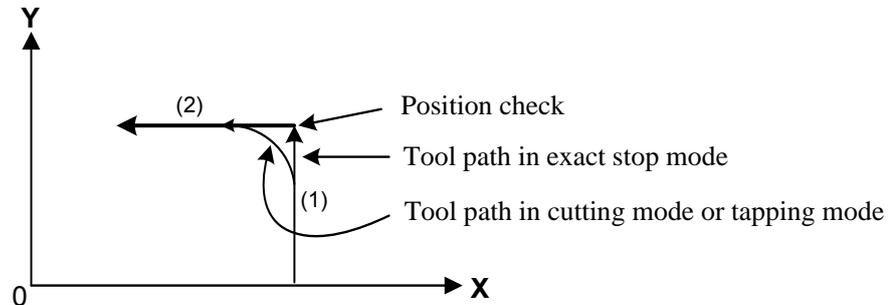


Fig. 4-6-1-1 Tool path from block 1 to block 2

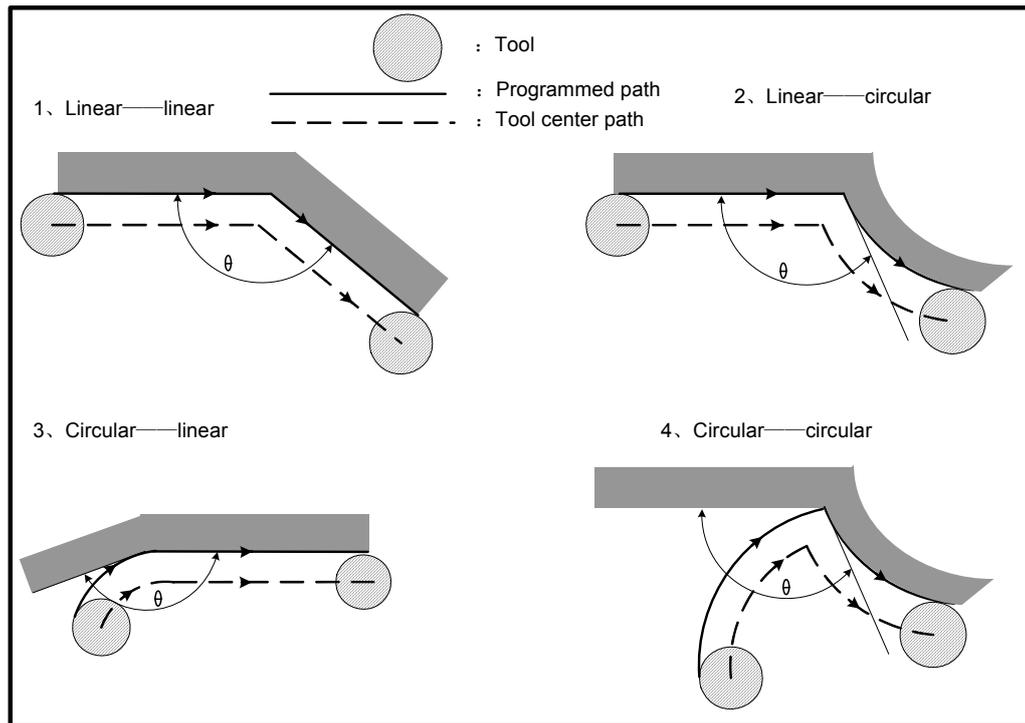
**4.6.2 Automatic override for inner corners (G62)**

**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、 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**

- 3、 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  $L_s + L_e \leq 2mm$ .

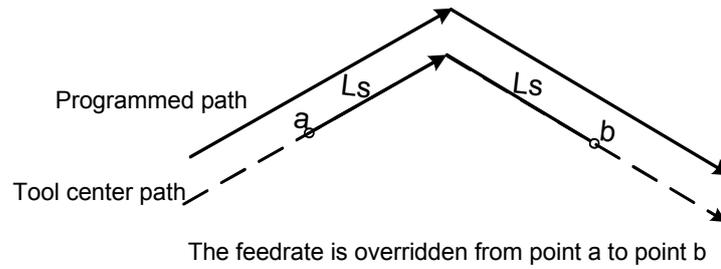


Fig. 4-6-2-2 Straight line to straight line

- 4 When a programmed path consists of two arcs, the feedrate is overridden if the start and end points are in the same quadrant or in adjacent quadrants. (Fig. 4-6-2-3)

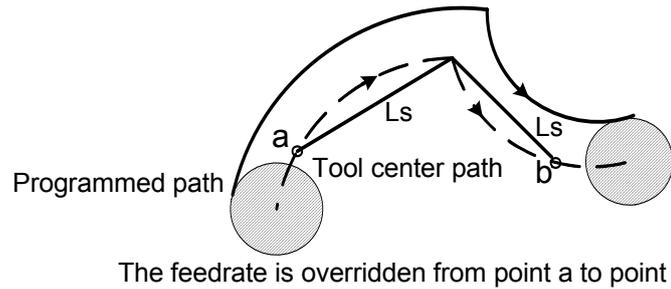


Fig. 4-6-2-3 Arc to arc

- 5 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)

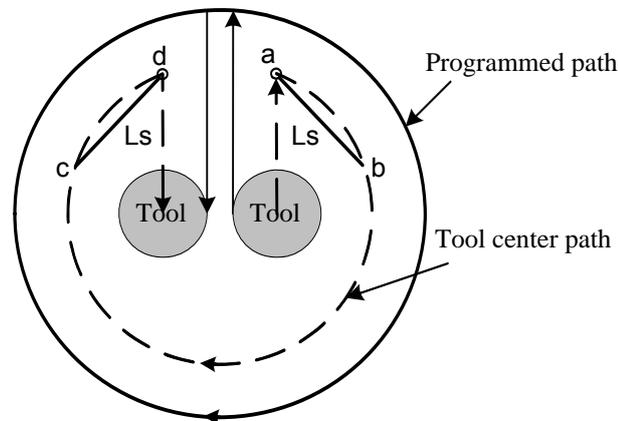


Fig. 4-6-2-4 Straight line to arc, arc to straight line

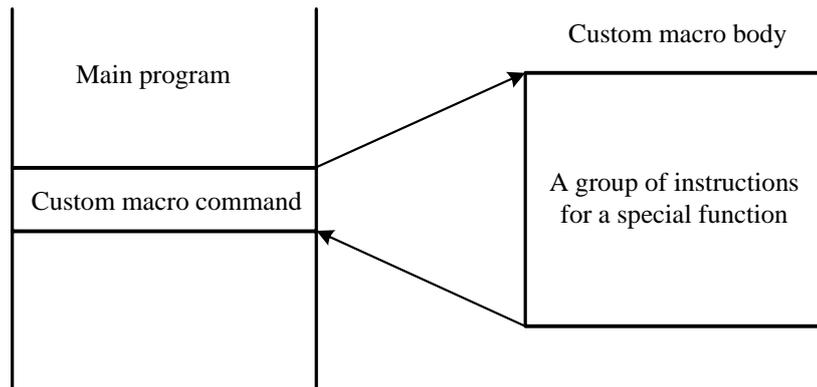
**Restriction**

- 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.

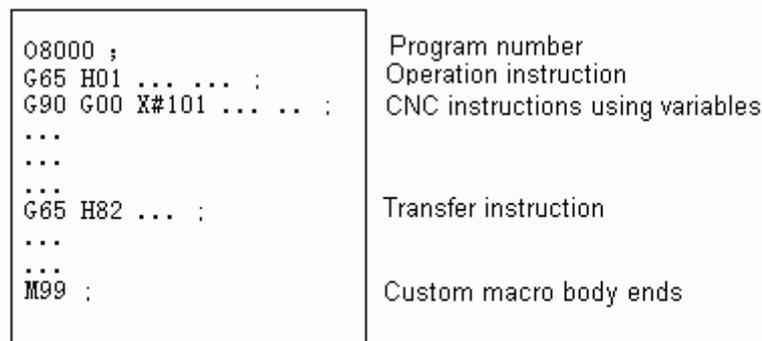


**Fig. 4-7-1-1**

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.



The composition of custom macro body

**Fig. 4-7-2-1**

#### 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.
- 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.
- 3** Display and setting of variable: It can be displayed on LCD, or be set by MDI.

**2. Types 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 is turned off or program end(operate M30, M02), bit parameter NO: 52#7 can be set whether to be cleared after resetting. When a macro is called, arguments are assigned to local variables.

(3) Common variables: #100~#199, #500~#999:

Bit parameter NO: 52#6 can determine whether the Common variables 100~#199 will be cleared.

Common variables usage not specified in this system can be used freely by user.

**Table 4-7-2-1**

| 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 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) Tool magazine data list #2700~#2955 (read-only, unwritable)
- 8) Alarm #3000
- 9) User data list #3500~#3755 (read-only, unwritable)
- 10) Modal message #4000~#4030 (read-only, unwritable)
- 11) Position message #5001~#5030 (read-only, unwritable)
- 12) Workpiece zero offset #5201~#5235 (readable and writable)
- 13) Additional workpiece coordinate system #7001~#7250 (readable and writable)

**3. Explanation for system variables**

- 1) Modal message

**Table 4-7-1-2**

| Variable 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 coordinate system )  |    |

- Note: 1 P code stands for the current selected additional coordinate system.**  
**2 When G#4002 code is being executed, the value obtained in #4002 is 17, 18 or 19.**  
**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       |                             |                          |  |

**Programming**

|       |  |                             |           |   |  |
|-------|--|-----------------------------|-----------|---|--|
|       | (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        | Unallowed                   |           |   |  |
| #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**

| <b>Variable No.</b> | <b>Function</b>                                      |
|---------------------|--|
| #5201               | External workpiece zero offset value of the 1st axis |
| ...                 | ...  |
| #5204               | External workpiece zero offset value of the 5th axis |
| #5206               | G54 workpiece zero offset value of the 1st axis      |
| ...                 | ...  |
| #5209               | G54 workpiece zero offset value of the 5th axis      |
| #5211               | G55 workpiece zero offset value of the 1st axis      |
| ...                 | ...  |
| #5215               | G55 workpiece zero offset value of the 5th axis      |
| #5216               | G56 workpiece zero offset value of the 1st axis      |
| ...                 | ...  |
| #5220               | G56 workpiece zero offset value of the 5th axis      |
| #5221               | G57 workpiece zero offset value of the 1st axis      |
| ...                 | ...  |
| #5225               | G57 workpiece zero offset value of the 5th axis      |
| #5226               | G58 workpiece zero offset value of the 1st axis      |
| ...                 | ...  |
| #5230               | G58 workpiece zero offset value of the 5th axis      |
| #5231               | G59 workpiece zero offset value of the 1st axis      |
| ...                 | ...  |
| #5235               | G59 workpiece zero offset value of the 5th axis      |
| #7001               | G54 P1 workpiece zero offset value of the 1st axis   |
| ...                 | ...  |
| #7005               | G54 P1 workpiece zero offset value of the 5th axis   |
| #7006               | G54 P2 workpiece zero offset value of the 1st axis   |
| ...                 | ...  |
| #7010               | G54 P2 workpiece zero offset value of the 5th axis   |
| #7246               | G54 P50 workpiece zero offset value of the 1st axis  |
| ...                 | ...  |
| #7250               | G54 P50 workpiece zero offset value of the 5th axis  |

**4. Local variable**

The correspondence of address and local variable:

Table 4-7-1-5

| <b>Argument address</b> | <b>Local variable No.</b> | <b>Argument address</b> | <b>Local variable No.</b> |
|-------------------------|---------------------------|-------------------------|---------------------------|
| A                       | #1                        | Q                       | #17                       |
| B                       | #2                        | R                       | #18                       |
| C                       | #3                        | S                       | #19                       |
| I                       | #4                        | T                       | #20                       |

**Programming**

|   |     |   |     |
|---|-----|---|-----|
| 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: -99999.9999~99999.9999

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

Example:

#100 = 35, #101 = 10, #102 = 5

#110 = #100÷#101 (=3.5)

#111 = #100×#102 (=17.5)

#120 = #100×#102 (=175)

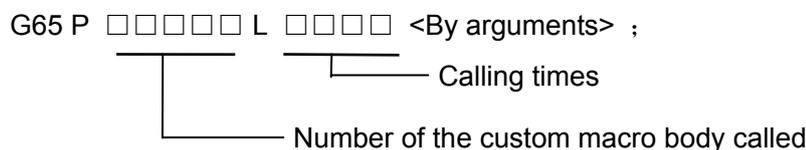
#121 = #120÷#101 (=17.5)

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 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 retrieved, 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.

### 4.7.4 Operation and transfer instruction

#### 4.7.4.1 Explanation for macro program 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 expressed directly without #.

#k: Variable name 2 for operation. Or a constant.

**Significance:** #i = #j ○ #k

○ Operation sign, specified by Hm

**Example:** P#100 Q#101 R#102.....#100 = #101 ○ #102 ;  
 P#100 Q#101 R15 .....#100 = #101 ○ 15 ;  
 P#100 Q-100 R#102.....#100 = -100 ○ #102 ;

H code specified by G65 has no effect 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 |

**Programming**

|     |     |                                      |                               |
|-----|-----|--------------------------------------|-------------------------------|
| G65 | H03 | Subtraction                          | #i = #j - #k                  |
| G65 | H04 | Multiplication                       | #i = #j × #k                  |
| G65 | H05 | Division                             | #i = #j ÷ #k                  |
| G65 | H11 | Logic addition(OR)                   | #i = #j OR #k                 |
| G65 | H12 | Logic multiplication (AND)           | #i = #j AND #k                |
| G65 | H13 | AND-OR                               | #i = #j XOR #k                |
| G65 | H21 | Square root                          | #i = $\sqrt{\#j}$             |
| G65 | H22 | Absolute value                       | #i =   #j                     |
| G65 | H23 | Compliment                           | #i = #j - trunc(#j ÷ #k) × #k |
| G65 | H24 | Algorism to binary                   | #i = BIN(#J)                  |
| G65 | H25 | Binary to algorism                   | #i = BCD(#J)                  |
| G65 | H26 | Compound multiplication and division | #i = (#i × #j) ÷ #k           |
| G65 | H27 | Compound square root                 | #i = $\sqrt{\#j^2 + \#k^2}$   |
| G65 | H31 | Sine                                 | #i = #j × SIN(#k)             |
| G65 | H32 | Cosine                               | #i = #j × COS(#k)             |
| G65 | H33 | Tangent                              | #i = #j × TAN(#k)             |
| G65 | H34 | 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: abandoning 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) 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 = ( # 101×# 102 ) ÷# 103)

13) 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 =  $\sqrt{\#102^2+\#103^2}$  )

14) 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 ))

15) 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))

16) 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))

17) 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.
  - 2 If Q, R required are not specified in operations above, they are defaulted for zero.
  - 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 ≠ #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 ≤ #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 ≥ #102, the execution proceeds by sequence.

6) Conditional transfer 5 #J.GE.# K ( ≥ )

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

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

When # 101 ≥ #102, it goes to N1000 block; when # 101 < #102, the execution proceeds by sequence.

7) Conditional transfer 6 #J.LE.# K ( ≤ )

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

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

When # 101 ≤ #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 #100.

#### 4. Logic AND, logic OR and logic NOT instructions

**Example:**

G65 H01 P#101 Q3;

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. Macro variable alarm

**Example:**

G65 H99 P1; macro variable 3001 alarm

G65 H99 P124; macro variable 3124 alarm

#### 4.7.4.2 Explanation for macro program B

##### 1. Arithmetic and logical operation

The following operation listed in table can be executed in variable. Right side expression of operation sign can contain constant and/or constant consist of function or operation sign. The variable #j and #k can be replaced by constant, the left side variable alos can be assigned value with expression.

**Table 4-7-4-2-1 arithmetic and logical operation**

| Function       | Format        | Remark |
|----------------|---------------|--------|
| Definition     | #i = #j       |        |
| Addition       | #i = #j + #k; |        |
| Subtraction    | #i = #j - #k; |        |
| Multiplication | #i = #j * #k; |        |

**Programming**

|                        |                       |   |
|------------------------|-----------------------|---|
| Division               | #i = #j / #k;         |   |
| Sine                   | #i = SIN[#j];         | Angle is specified by degree , 90°30' means 90.5 degree |
| Arcsine                | #i = ASIN[#j];        |   |
| Cosine                 | #i = COS[#j];         |   |
| Ant cosine             | #i = ACOS[#j];        |   |
| Tangent                | #i = TAN[#j];         |   |
| Arc tangent            | #i = ATAN[#j] / [#k]; |   |
| Square root            | #i = SQRT[#j];        |   |
| Absolute value         | #i = ABS[#j];         |   |
| Round off              | #i = ROUND[#j];       |   |
| Top integral           | #i = FIX[#j];         |   |
| FUP                    | #i = FUP[#j];         |   |
| Natural logarithm      | #i = LN[#j];          |   |
| Exponential function   | #i = EXP[#j];         |   |
| OR                     | #i = #j OR #k;        | logical operation is executed by binary bit by bit      |
| XOR                    | #i = #j XOR #k;       |   |
| AND                    | #i = #j AND #k;       |   |
| Switch from BCD to BIN | #i = BIN[#j];         | Used for exchanging signal with PLC                     |
| Switch from BIN to BCD | #i = BCD[#j];         |   |

**Explanation:**

➤ Angle unit

The unit of function SIN, COS, ASIN, ACOS, TAN and ATAN is degree. Such as 90° 30 ´ means 90.5 degree.

➤ ARCTAN

Specify lengt of two sides and part with sprit ( / ).

➤ Abbreviation for operation and logical operation instruction:

When specifying function in program, former two signs of function name can be used to specify the function.

example:

ROUND→RO

FIX→FI

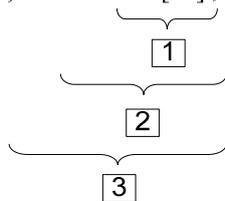
➤ Operation sequence

①function

②Multiplication and Division operation\* / AND)

③Addition and Subtraction operation ( + - OR XOR )

Example )  $2 + \#3 * \text{SIN}[\#4] ;$



1、 2 and 3 means operation sequence.

## 2. Transfer and cycle

### 1) transfer and cycle

Flow direction of control can be changed with GOTO and IF statement in program. There are three kinds of transfer and cycle operations can be used.



### 2) Nnconditional transfer

#### ➤ GOTO statement

Transfer to block with sequence number n. sequence number can be specified by expression

GOTOn; n: sequence number (1 to 99999)

Example:

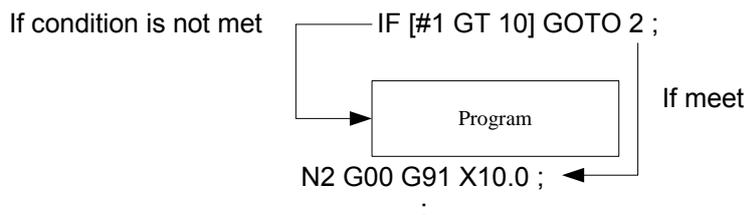
GOTO 1;  
GOTO #10;

### 3) conditional transfer (IF statement) [ <condition expression > ]

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

If the expression meets the specified condition, transfer to the block with sequence n; if not, the next block will be executed.

If variable #1 value is more than 10,  
transfer to block with sequence number N2.



**IF[<condition expression >]THEN**

**Programming**

If meet the condtion, preset macro statement will be executed. Only one macroco can be executed.

|   |
|---|
| If the value of #1 and #2 is the same, 0 assign value to 3. |
| IF[#1 EQ #2] THEN #3=0;                                     |

**Explanation:**

- Conditional expression

It must contain operator. Operator is placed between two variables or between varable and constant, and then seal off with ([ , ]). Expression can replace variable.

- Operator

It consists of two letters, which is used for comparing two values to determine whether they are equal or not equal.

**Table 4-7-4-2-2 operator**

| Operator | Meaning                   |
|----------|---------------------------|
| EQ       | Equal to (=)              |
| NE       | Unequal to (≠)            |
| GT       | More than (>)             |
| GE       | More than or equal to (≥) |
| LT       | Less than (<)             |
| LE       | Less than or equal to (≤) |

- Typical program

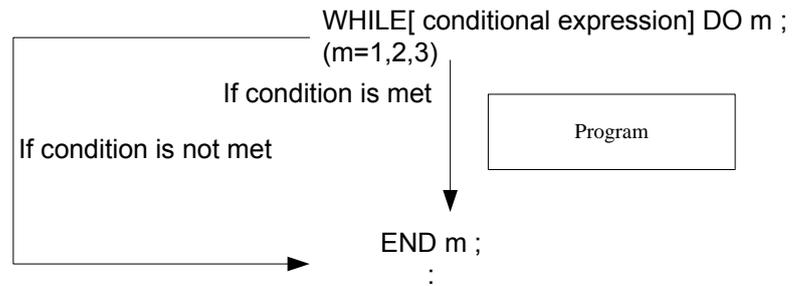
The following program is used for caculating the summation of 1~10.

```

O9500;
#1=0; .....Initial value of sum of
                storage
#2=1; .....Initial value of augend
                variable
N1 IF[#2 GT 10]GOTO 2; .....When augend is more than
                10, transfer to N2.
#1=#1+#2; .....summation
#2=#2+#1; .....Next augend
GOTO 1; .....Transfer to N1
N2 M30; .....End
    
```

4) Cycle (WHILE statement )

Specify a expression follow WHILE statement, when the condition is met, execute the program between DO and END, otherwise, transfer to a block behind END.



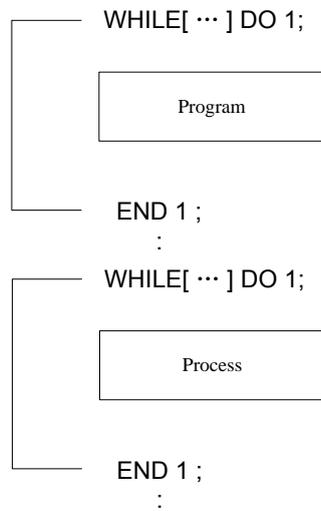
When specified condition is met, execute the program between DO and END, otherwise, transfer to a block behind END. This instruction format is suitable for IF statement, number behind DO and END is the sign used to specify program execution range. Sign value is for 1, 2, 3. if the value beyond 1, 2, 3 is used, alarm occurred.

**Explanation:**

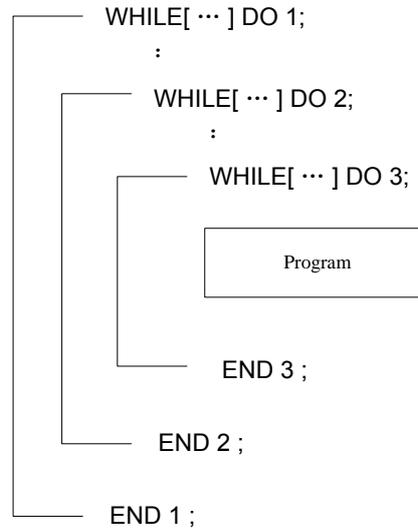
➤ Nesting

Sign (1 to 3 ) in DO—END cycle can be used repeatedly according to requirement. When the DO range superimpose, alarm occurred.

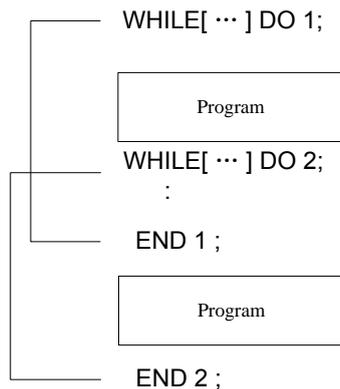
1. Sign (1 to 3 ) in DO—END Cycle can be used repeatedly according to requirement



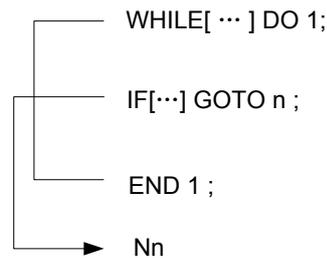
3. DO cycle can nest three levels



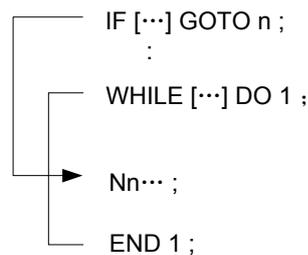
2. Range of DO can not superimpose



4. control can transfer outside the cycle.



5. transfer can not enter into cycle



➤ Typical program

The following program is used for calculating the summation of 1~10.

```

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

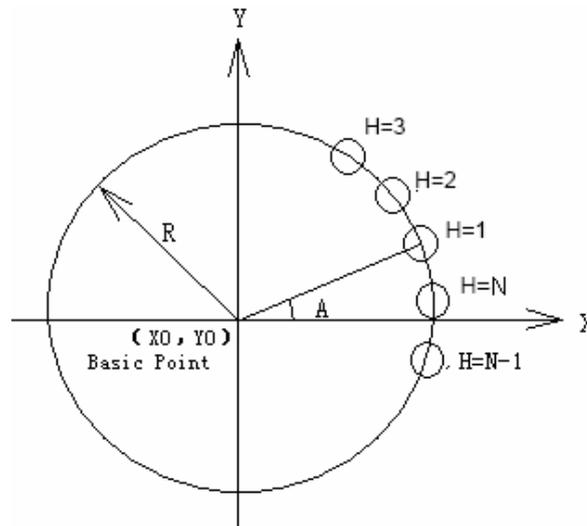
**Precautions:**

- When using M, S, T, D,F to lead variable in G65 calling macro program, only positive integer is valid, and the others letter without the limitation.
- Code with line number N can not in a same line with WHILE/DO/END, otherwise, alarm occurred in jumping.
- Cycle and skip instruction is invalid in DNC.
- Search begin from GOTO statement heading, when the 1st corresponding line number is found, program jump. Try your best not to use a same N code in a block.
- When express variable number with decimal, system directly rounding decimal part, and will not consider carry digit.
- Local variable value will keep till main program end, and it is used together in each subprogram.

**4.7.5 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

**Programming**

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 Q0;           I=0
G65 H22 P#101 Q#504;           IE= | N |
N200 G65 H04 P#102 Q#100 R3600 00;
G65 H05 P#102 Q#102 R#504;      $\theta_i = A + 360^\circ \times I/N$ 
G65 H02 P#102 Q#503 R#102;
G65 H32 P#103 Q#502 R#102;      $X_i = X_i + R \cdot \cos(\theta_i)$ 
G65 H02 P#103 Q#500 R#103;
G65 H31 P#104 Q#502 R#102;      $Y_i = Y_i + R \cdot \sin(\theta_i)$ 
G65 H02 P#104 Q#501 R#104;
G90 G00 X#103 Y#104;           The i-th hole positioning
M10;                           Hole drilling M code output.
G65 H02 P#100 Q#100 R1;        i=i+1
G65 H84 P200 Q#100 R#101;     When i < IE, it goes to N200 to drill IE holes.
M99;
```

Program examples for calling custom macro body is as following:

```
O0010;
G65 H01 P#500 Q100000;         X0=100MM
G65 H01 P#501 Q-200000;       Y0=-200MM
G65 H01 P#502 Q100000;       R=100MM
G65 H01 P#503 Q20000;        A=20°
G65 H01 P#504 Q12;          N=12 in CCW direction
G92 X0 Y0 Z0;
M98 P9010;                   Calling custom macro
G80;
X0 Y0;
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  |
|                                  | M20                  | Spindle gap instruction  |
|                                  | M21                  | Tool search instruction in retraction                                |
|                                  | M22                  | Tool search instruction for a new tool                               |
|                                  | M23                  | Magazine to spindle instruction                                      |
|                                  | M24                  | Magazine retraction instruction                                      |
|                                  | M26                  | Start chip remove hydrovalve   |
|                                  | M27                  | Close chip remove hydrovalve   |
| M28                              | Cancel rigid tapping |  |
| M29                              | Rigid tapping        |  |
| M32                              | Lubricating on       |  |

**Programming**

|  |     |                                |
|--|-----|--------------------------------|
|  | M33 | Lubricating off                |
|  | 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 | Check if spindle has a tool    |

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)              It 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 Chip remove hydrovalve ON/OFF (M26, M27)**

Instruction: M26 chip remove hydrovalve ON; M27 chip remove hydrovalve OFF.

**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.

**5.1.16 Check if spindle has a tool (M55)**

Instruction: M55, check if it is right after 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 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 control returns to the beginning of the program while the numbers of the workpiece machined added by 1. Under any situations M30 (M02) is regarded as the end of the program execution. 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.

### 5.2.2 Program dwell (M00)

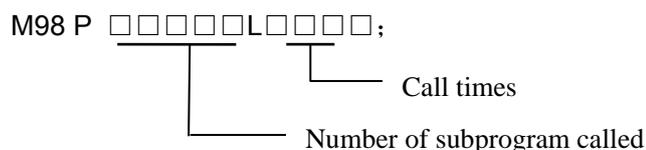
In Auto running, automatic operation pauses after a block containing M00 is executed. And the previous modal information will be saved. The automatic operation can be continued by pressing cycle start key, which is equivalent to pressing down feed hold key.

### 5.2.3 Program optional stop (M01)

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

### 5.2.4 Subprogram calling (M98)

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



### 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. M99 is taken as M30 in DNCmode, and cursor will place in program ending.

## 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 Alarm is issued and the execution stops if S code beyond the codes above is specified in program.
- 2 For a two-digit S code, the latter two digits are effective if S code is specified with a four-digit number.

### 6.3 Constant surface speed control (G96/G97)

**Format:**

Constant surface speed control instruction

G96 S\_ Surface speed (m/min or feet/min)

Constant surface speed control cancel instruction

G97 S\_ Spindle speed (rpm)

Constant surface speed controlled axis instruction

G96 Pn P1 X axis; P2 Y axis; P3 Z axis; P4 4<sup>th</sup> axis

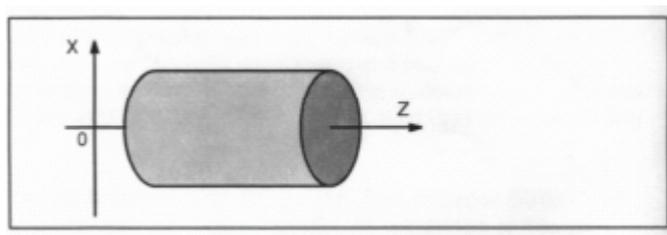
Max. spindle speed clamping

G92 S\_ S specifies the max. spindle speed (rpm)

**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:**

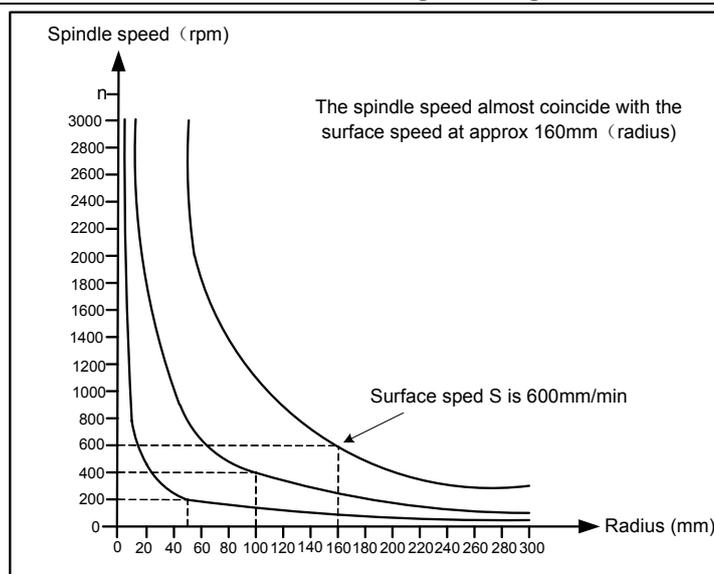
- 1 G96 is a modal instruction. After it is specified, the program enters the constant surface speed control mode and the S value specified is assumed as a surface speed.
- 2 A G96 instruction must specify the axis along which constant surface speed control is applied. It can be cancelled by G97 instruction.
- 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**

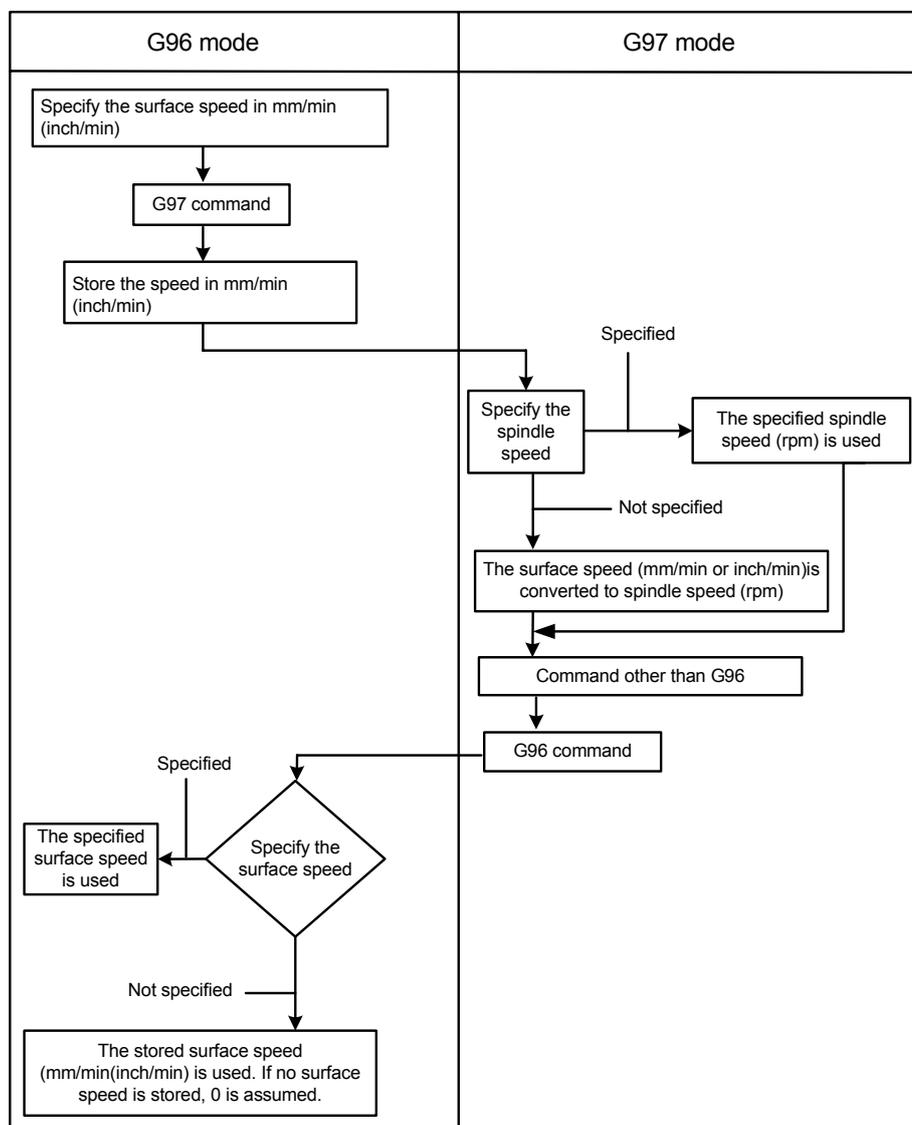
- 4 When constant surface speed control is applied, if a spindle speed higher than the value specified in G 92 S\_, it is clamped at the maximum spindle speed. When the power is switched on, the maximum spindle speed is not yet set, the S in G96 is regarded as zero till M3 or M4 appears in program.

**Programming**



**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:

|  |   |   |  |
|--|---|---|--|
|  F0 |  25% |  50% |  100% |
| 0.001  | 0.01  | 0.1   | 1  |

In which, F0 is set by number parameter P93.

The acceleration of rapid positioning (G0) can be set by number parameter P105~123. 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:

- 1、 Feed per minute (G94): it is used to specify the feed amount per minute after F code.
- 2、 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.
- 3、 An override from 0% to 150% can be applied to feed per minute with the override key on the operator panel.

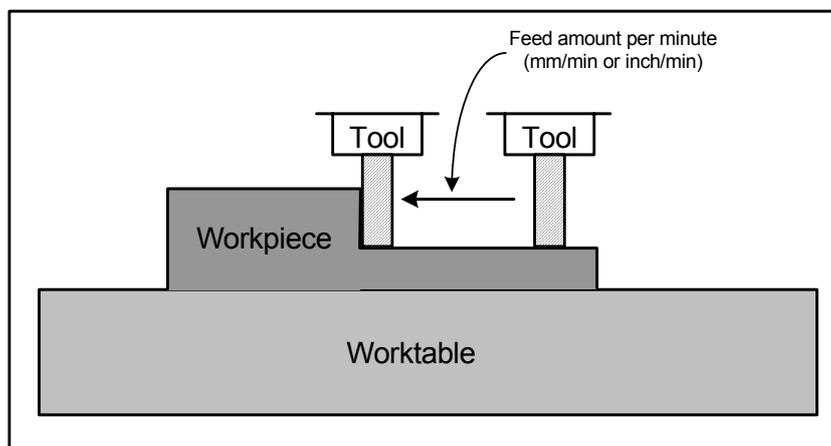


Fig. 7-2-1-1 Feed per minute

**Restriction**

Feed per minute mode can't be applied for some instructions such as threading.

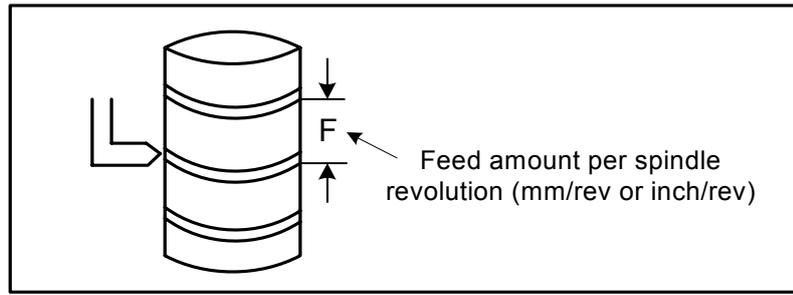
### 7.2.2 Feed per revolution (G95)

**Format:** G95 F\_

**Function:** It specifies the tool feed amount in a revolution. Unit: mm/rev or inch/rev

**Explanation:**

- 1 After specifying G95 (feed per revolution mode), the feed amount of the tool per revolution is directly specified by a number after F.
- 2 G95 is a modal code. Once specified, it is effective till G94 is specified.
- 3 An override from 0% to 150% can be applied to feed per revolution with the override key on the operator panel.

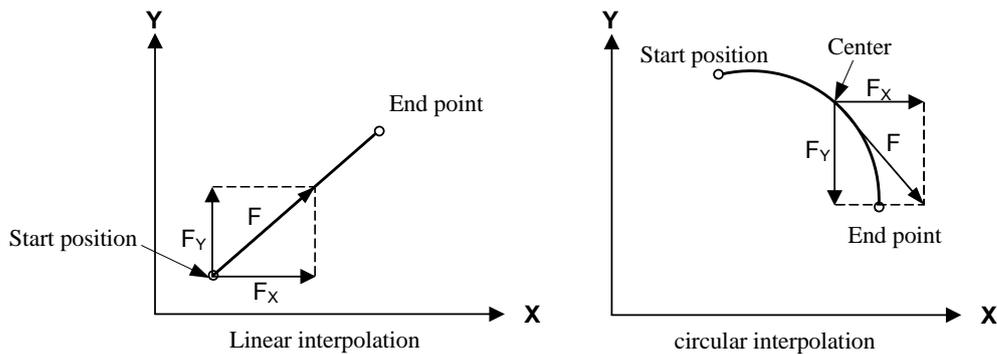


**Fig. 7-2-2-1 Feed per revolution**

**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 MANUAL 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)

MANUAL feed: Hind acceleration/deceleration (0: linear type; 1: exponential type)

(Set the universal time constant for each axis by parameters)

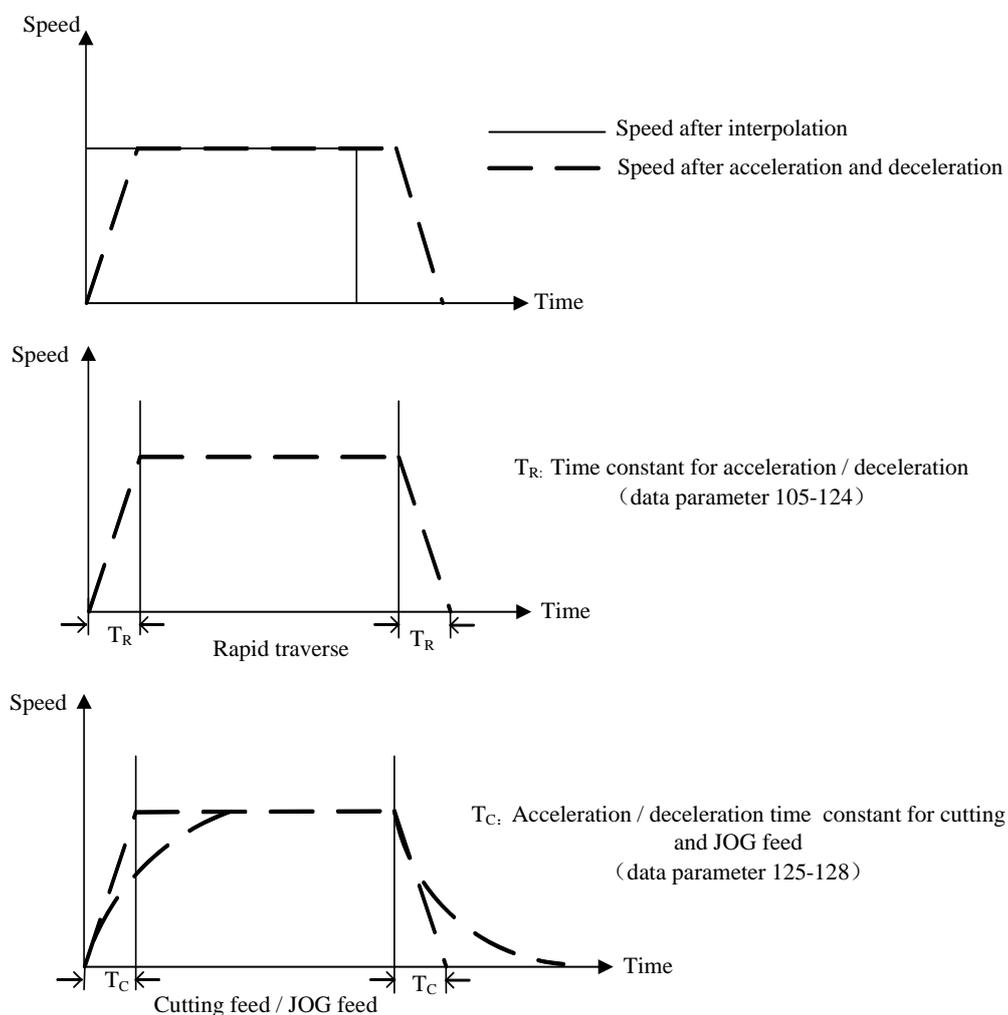
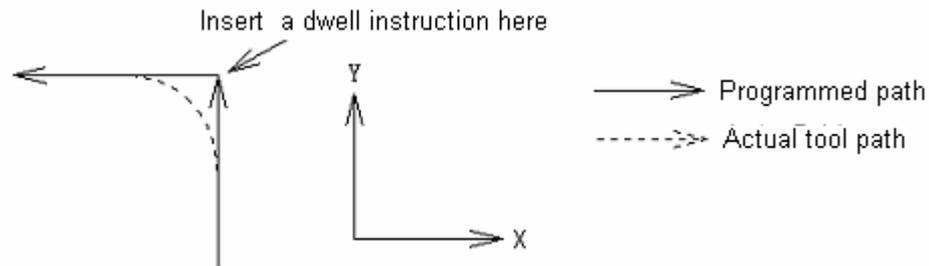


Fig. 7-5-1

## 7.6 Acceleration/deceleration for corner of a block

For example: if 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. 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

# 1 Operator Panel

## 1.1 Panel layout

The 218M machine center has an integrated operator panel, which is comprised by LCD area, edit area, interface display area and machine control area. The layout of it is shown as following:

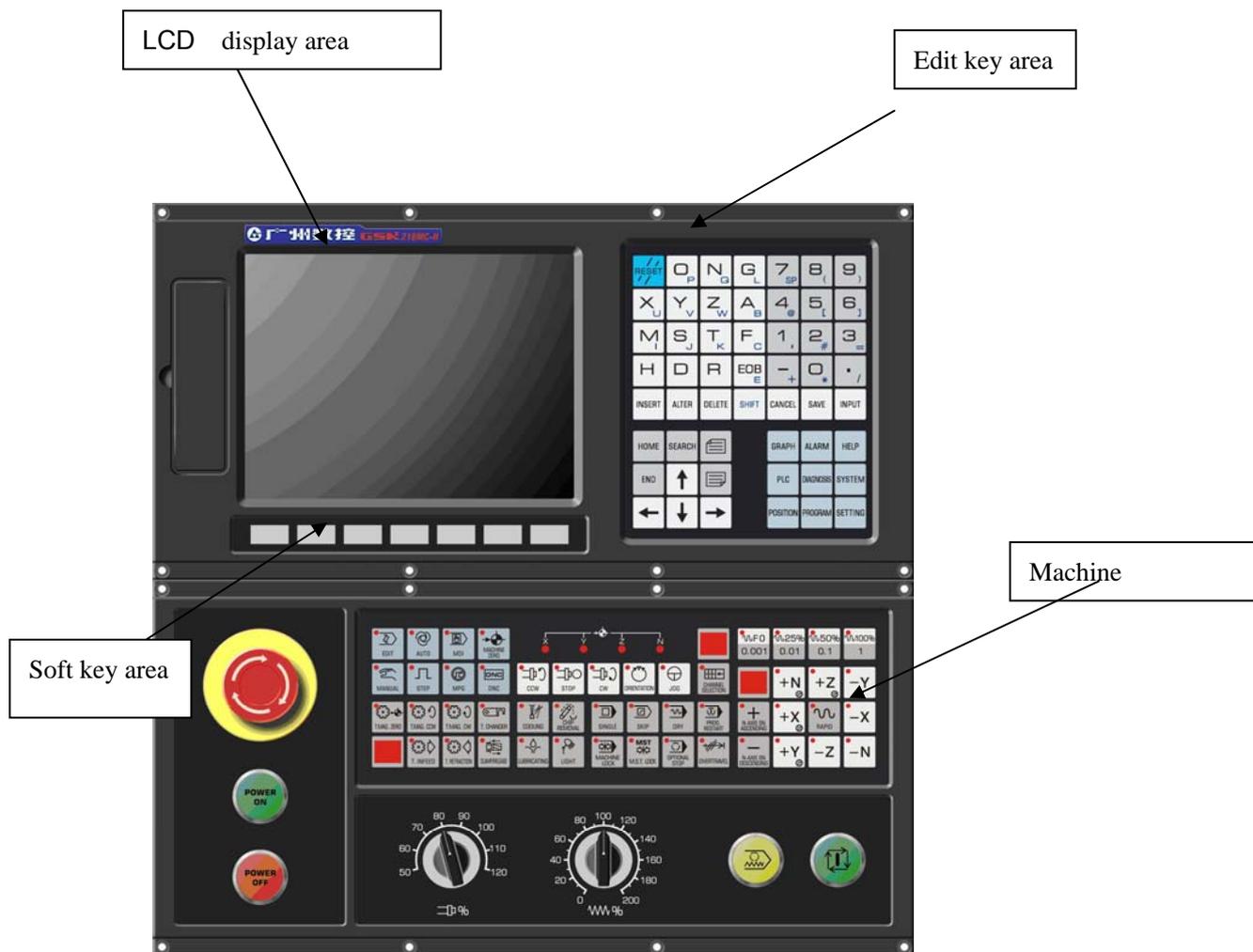


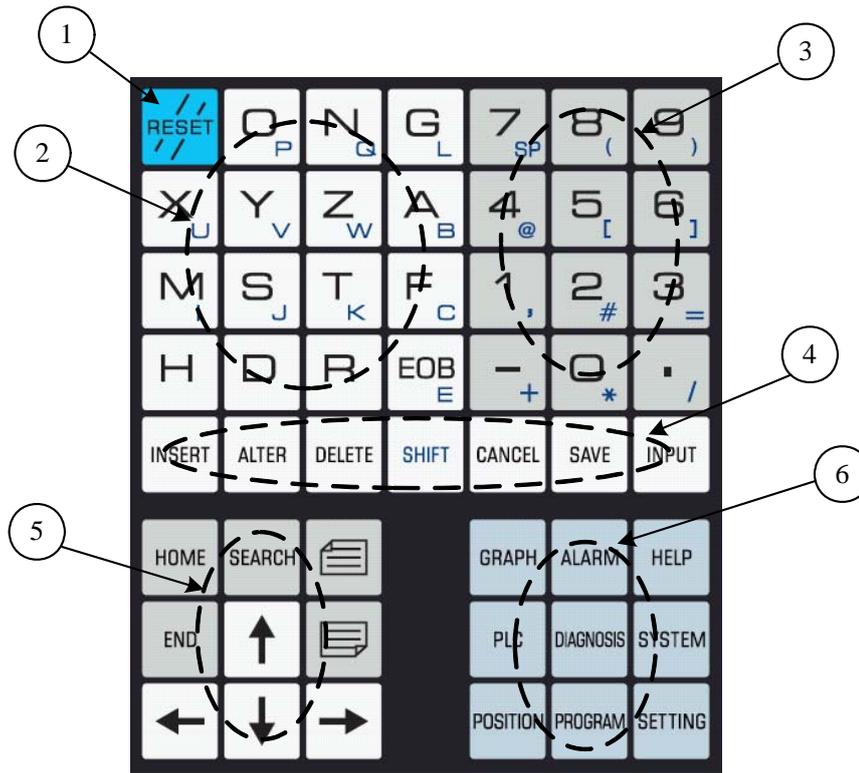
Fig. 1-1-1 panel layout

## 1.2 Explanation of the panel function

### 1.2.1 LCD area

The display area of this system is applied with a 10.4 inch chromatic LCD that has 640×480 resolution.

**1.2.2 Edit area**



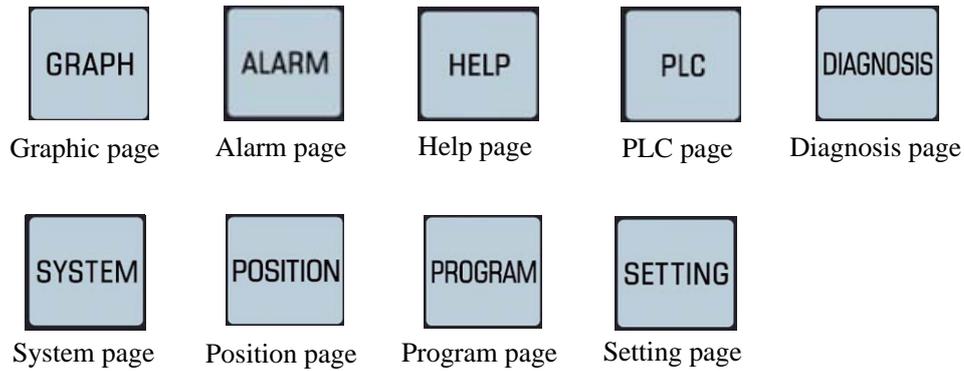
**Fig.1-2-2-1 edit area**

The edit keyboard divided for 12 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      | Edit keys             | Press these keys to insert, modify and delete the programs, words etc. in program edit |
| 5      | Cursor keys           | Press these keys to move the cursor in four directions                                 |
| 6      | Screen operation keys | Press any key of them to enter the corresponding interface (introduced below)          |

**1.2.3 Screen operation keys**

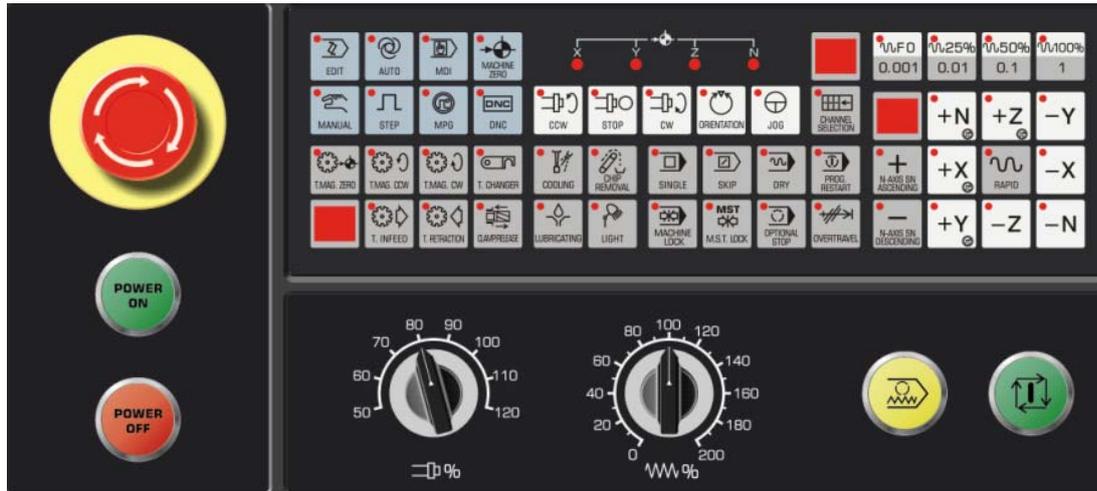
8 page display keys and 1 help display key are laid out in this operator panel, which are as following:



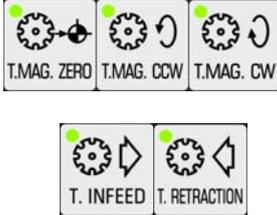
| Name           | Explanation                            | Remark  |
|----------------|--|---|
| Graphic page   | Press this key to enter setting page   | For graphic parameter, graphic display page and the graphic center, dimension, ratio and display page setting by changing soft keys           |
| Alarm page     | Press this key to enter alarm page     | To view alarm display pages by changing soft keys   |
| Help page      | Press this key to enter help page      | To view the corresponding help message by changing soft keys  |
| PLC page       | Press this key to enter PMC page       | To view the corresponding system version, the I/O configuration and modify PLC ladder in Edit mode by changing soft keys                      |
| Diagnosis page | Press this key to enter diagnosis page | To view the I/O signals in the system by changing soft keys   |
| System page    | Press this key to enter system page    | To view tool offset, parameter, macro variable and screw page by changing soft keys   |
| 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, current/time, program directory display by changing soft keys, and program directory is switched over by page keys |
| Setting page   | Press this key to enter program page   | 2 pages, setting, parameter switch, coordinate, panel, servo system, data and password setting display by changing soft keys                  |

**Note** The pages by soft keys above can also be displayed by continuously pressing the corresponding functional keys by setting bit parameter NO:25#0~25#7、NO:26#6~26#7 . Refer to Chapter 3 in this manual for the details of these pages.

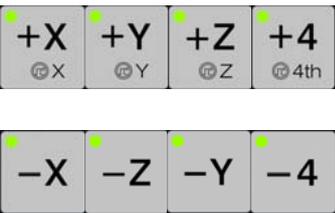
**1.2.4 Control area**



| Keys  | Name                  | Function explanation       | Remarks and operation explanation   |
|---|-----------------------|----------------------------|---|
|   | Edit mode 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          |
|  | Step mode key         | To enter Step mode         | Switching to Step 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                   |
|  | MPG mode key          | To enter MPG mode          | Switching to MPG 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 |
|    | Spindle control keys           | Spindle CCW<br>Spindle stop<br>Spindle CW  | MPG mode, Step mode, MANUAL mode   |
|    | Spindle override keys          | Spindle speed adjusting (spindle speed analog control effective)                         | Any mode   |
|    | Spindle JOG key                | Spindle JOG on/off   | MANUAL mode, STEP mode, MPG mode   |
|    | Spindle Orientation key        | Spindle orientation on/off   | MANUAL mode  |
|   | Tool magazine operation keys   | Tool magazine operation on/off   | MANUAL mode  |
|  | Manual tool clamp/ release key | For manual tool clamping / release   | MANUAL mode  |
|  | Manual tool change key         | For manual tool change   | MANUAL 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  |
|  | Single Block key               | For switching of program block/blocks, if it is 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                    | If the   | Auto mode, MDI mode,   |

**Operation**

|   |                                    |   |  |
|---|------------------------------------|---|--|
|    | key                                | miscellaneous function is on, its indicator lights up and M,S,T functions are ineffective   | 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, MANUAL mode, DNC mode            |
|    | Working Light key                  | For machine working light on/off  | Any mode   |
|    | Lubricating key                    | For machine lubrication on/off  | Any mode   |
|    | Cooling key                        | For coolant on/off  | Any mode   |
|   | Chip Removal key                   | Chip removal switch   | Any mode   |
|  | Feedrate Override keys             | For adjustment of the feedrate  | Auto mode, MDI mode, Edit mode, Machine zero, MPG mode, Step mode, MANUAL mode, DNC mode |
|  | Rapid traverse key                 | For rapid traverse on/off   | Any mode   |
|  | Rapid, Step, and MPG override keys | For rapid override, manual single step, MPG override selection  | Auto mode, MDI mode, Edit mode, Machine zero, MPG mode, Step mode, MANUAL mode, DNC mode |
|  | Manual feeding keys                | For positive/negative moving of X, Y, Z 4 <sup>th</sup> axis in MANUAL, Step mode, and the positive direction of the axis is by MPG | Machine zero, MPG mode, Step mode, MANUAL mode   |

|  |                               |  |                                      |
|--|-------------------------------|--|--------------------------------------|
|  <p>OVERTRAVEL</p>    | <p>Overtravel release key</p> | <p>Alarm occurring if machine reaches the hard limit, pressing this key with indicator lighting up to move reversely till the indicator gone out</p> | <p>MANUAL mode, MPG mode</p>         |
|  <p>PROG. RESTART</p> | <p>Program Restart key</p>    | <p>Cursor moving to the beginning of the starting block to restart the machine, also for rapid program check</p>                                     | <p>Auto mode, MDI mode, DNC mode</p> |
|  <p>OPTIONAL STOP</p> | <p>Optional Stop key</p>      | <p>For stop of the program with "M01"</p>  | <p>Auto mode, MDI mode, DNC mode</p> |
|  <p>FEED HOLD</p>    | <p>Feed Hold key</p>          | <p>Auto running stops by pressing this key</p>   | <p>Auto mode, MDI mode, DNC mode</p> |
|  <p>CYCLE START</p> | <p>Cycle Start key</p>        | <p>Auto running begins by pressing this key</p>  | <p>Auto mode, MDI mode, DNC mode</p> |

## 2 System Power On/Off and Safety Operations

### 2.1 System power on

Before this GSK218MC-H 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.

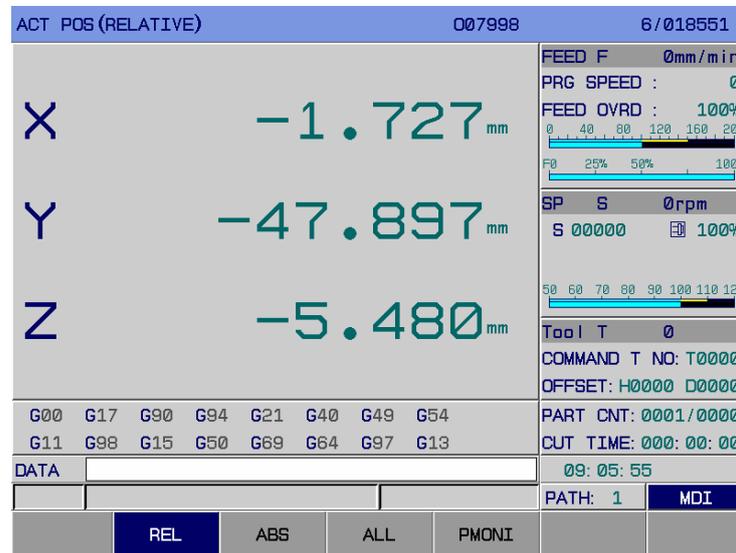


Fig. 2-1-1

### 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 N35.1~N35.7 and N36.0~N36.7 and resetting;
- 4、 Whether to clear F, H, D codes or not after modifying bit parameters N34.7 and resetting;
- 5、 Whether to delete the program or not after modifying bit parameters N28.7 and resetting in MDI mode;
- 6、 Whether to clear the executing DNC program display or not after modifying bit parameters N23.2 and resetting;
- 7、 Whether to cancel local coordinate or not after modifying bit parameters N10.3 and resetting;
- 8、 Whether to clear the macro common variables #0-#99 or not after modifying bit parameters N52.6 and resetting;
- 9、 It can be used in system abnormal output and coordinate axis action.

### 2.3.2 Emergency stop

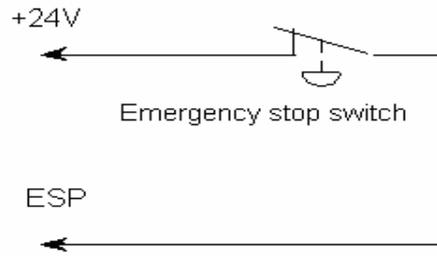
If the emergency button is pressed during machine running, the system enters into emergency status and the machining movement is stopped immediately. All the outputs such as the spindle running, coolant are also cut off. If the emergency button is released (varying by machine builders, usually the button 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.

- 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:

**Operation**



**Fig. 2-3-2-1**

**2.3.3 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 and 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 modifying the address K5.1 of the PLC.

**2.5 Overtravel protection**

Overtravel protection must be employed to prevent the damage to the machine due to the overtravel of the X, Y, or Z axis.

**2.5.1 Hardware overtravel protection**

The overtravel limit switches are fixed at the positive and negative maximum stroke of the machine X, Y, 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.

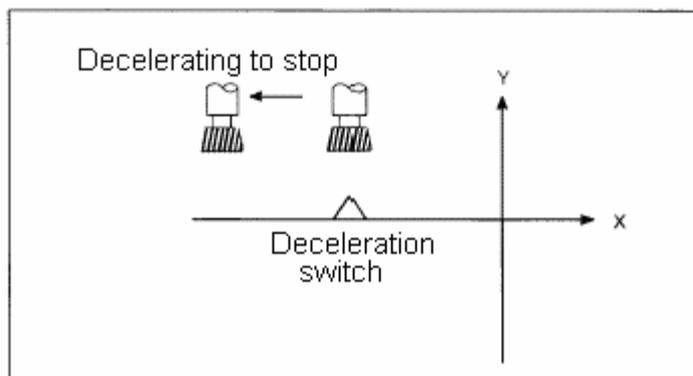


Fig. 2-5-1-1

**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 MANUAL mode**

In MANUAL 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 NO.66~NO.73 (appendix 1), referring to machine coordinate values.

Overtravel alarm occurs if the machine position (coordinate) exceeds the setting software stroke. The alarm issued before or after overtravel for software limit overtravel is set by bit parameter No.11.7. During the overtravel alarm, move the axis reversely in MANUAL mode, the alarm will be cancelled after the axis is moved out of the overtravel range.

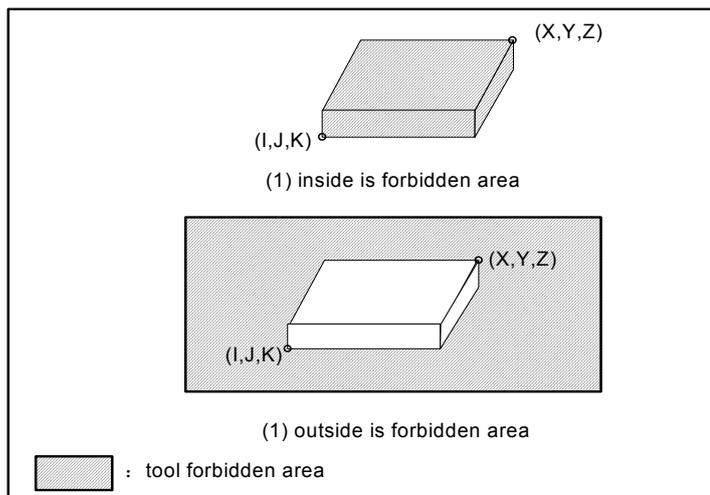
**2.5.3 Release of the overtravel alarm**

The method for overtravel alarm is: in MANUAL mode or MPG mode, press the <OVERTRAVEL> key on the panel, then move the axis out reversely (for positive overtravel, move negatively; for negative overtravel, move positively).

**2.6 Storage stroke detection**

**Operation**

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**

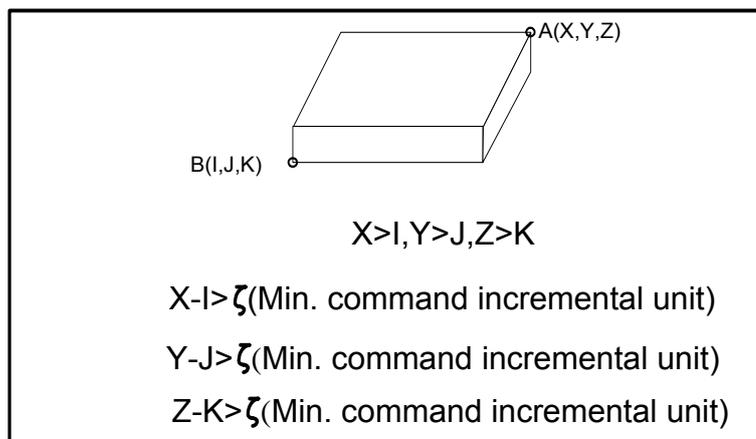
When the tool goes beyond the stroke, alarm is issued and the machine decelerates and stops.

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~P73, 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~P83 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.



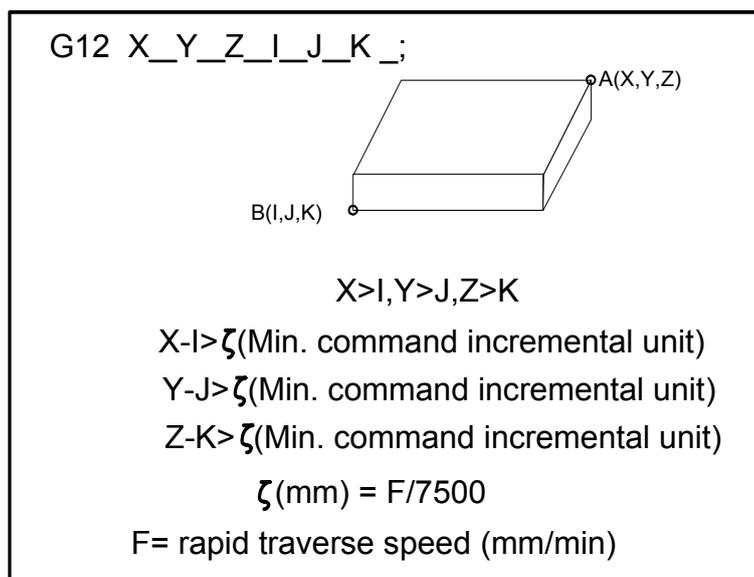
**Fig. 2-6-2 To use parameters set up or change forbidden area**

Even the coordinate sequence of the 2 points is wrongly given in storage stroke detection 2, a rectangular forbidden area can also be formed by these 2 points

taken as the vertex.

When the forbidden area is set by number parameter P76~P83, 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.

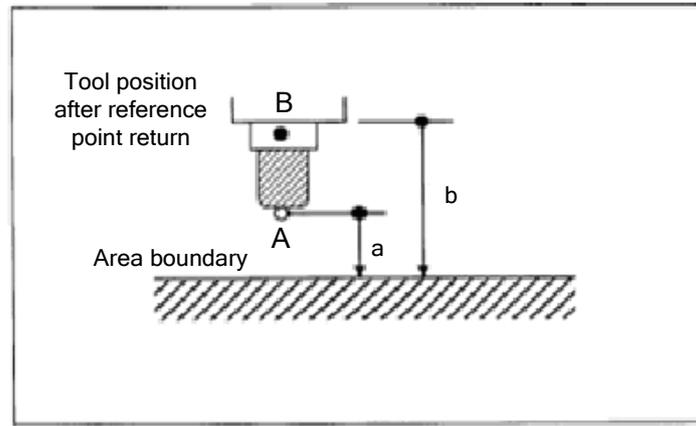


**Fig.2-6-3 To set up or change forbidden area by program**

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 used by the parameter.

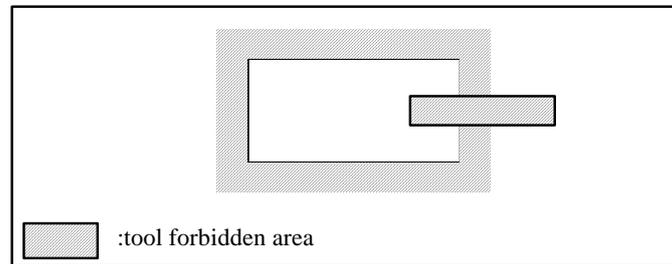
3. Detection point in the forbidden area: prior to the programming for forbidden area, please confirm the detection point (the top of the tool nose or tool collet). As is shown in Fig.2-6-4, if the detection point is A (tool nose), the distance "a" should be used as the data for storage detection; if 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.

**Operation**



**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:

### 3 Interface Display as well as 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 by pressing **【REL】** soft key, called “relative” in following (see Fig.3-1-1-1):

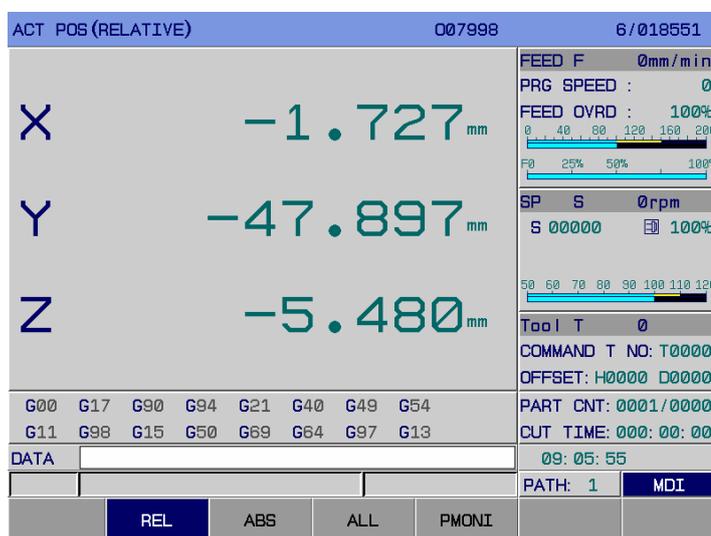
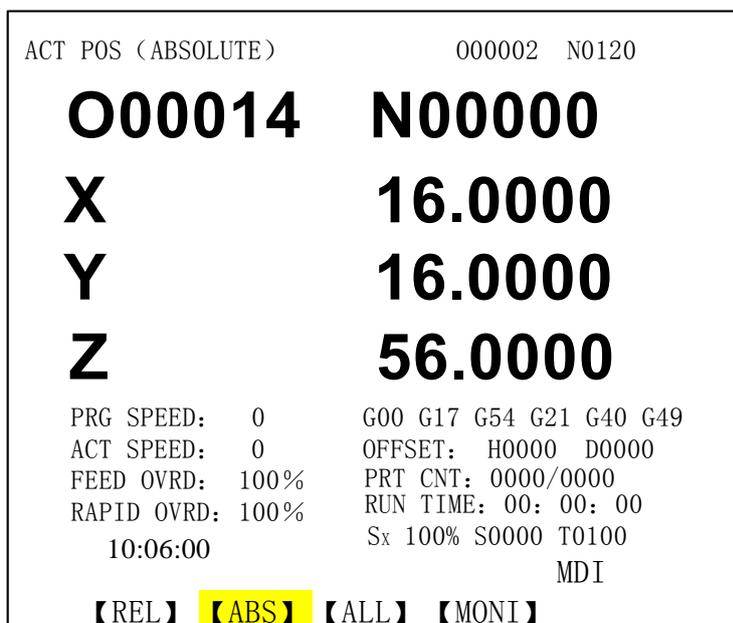


Fig. 3-1-1-1

Clearing steps of relative coordinate system: press <X> key, as X axis blinks, press <CANCEL> key to clear the coordinate system, and the same for Y and Z axes.

Setup steps of relative coordinate system: press <X> key, as X axis blinks, input the setting data then press <ENTER> key to enter the data into coordinate system.

2) Absolute mode: It displays the absolute coordinate of the current tool by pressing **【ABS】** soft key, which is called “absolute” in following(see Fig.3-1-1-2):



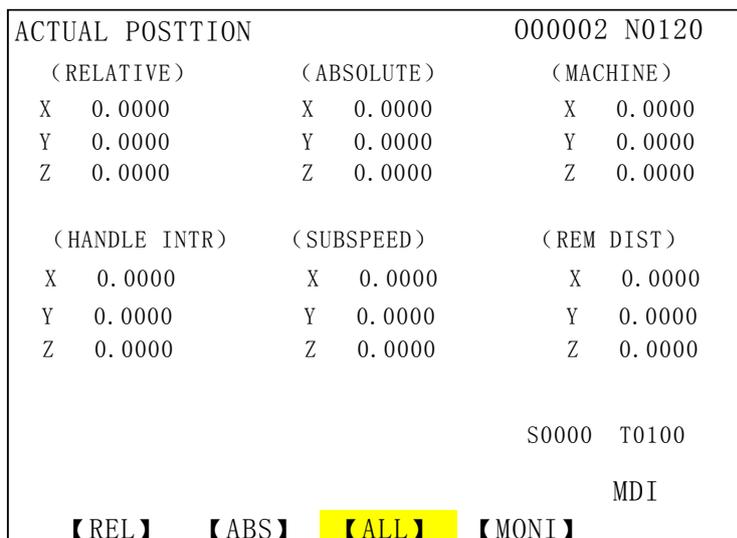
**Fig. 3-1-1-2**

**3) All mode**

It enters **【ALL】** mode by pressing **【ALL】** soft key, 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) :



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

4) Monitoring mode

It enters 【MONI】 mode by pressing 【MONI】 soft key, in this mode 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)

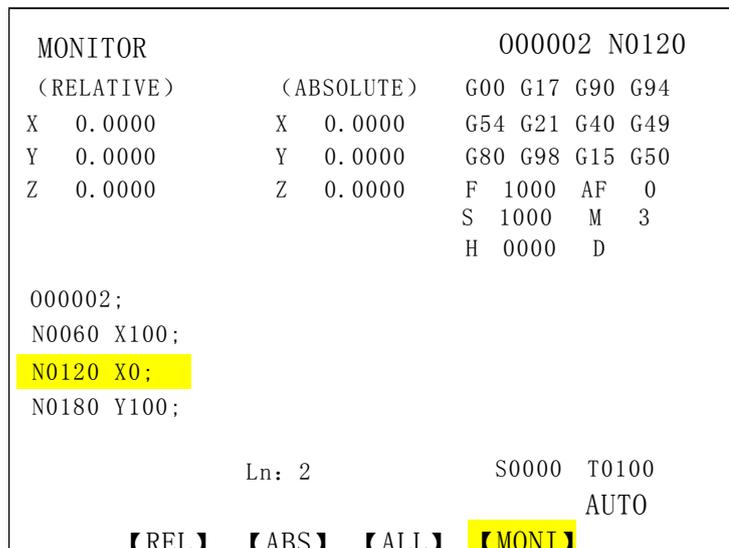
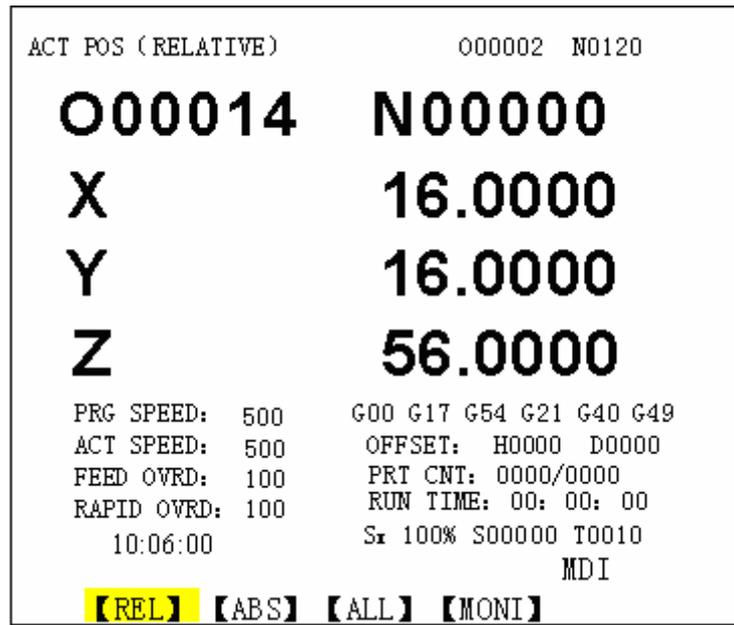


Fig. 3-1-1-4

- 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.
- 2** In <MACHINE ZERO>, <STEP>,<MANUAL>,<MPG>modes, the intermediate coordinate system is a relative one; while in <AUTO>,<MDI>,<DNC> modes, it is a remaining distance.

**3.1.2 The display of the 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 number, spindle override, spindle speed, tools etc. can be displayed in <POSITION> absolute or relative mode(see Fig.3-1-2-1).



**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 if Auto run starts, time units are hour, minute and second

S<sub>x</sub> : 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:

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, 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 No.088~093 and they can be overridden by rapid override; the dry run speed is set by number parameter No.086.
- 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 P355, total workpieces to be machined is set by number parameter P356.

### 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. 3-1-2-1)

- 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. 3-1-5)

- 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.

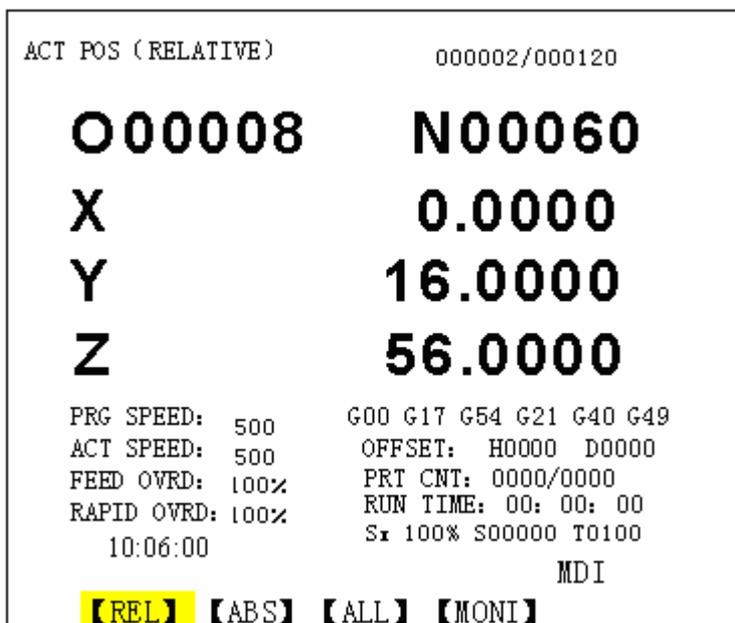


Fig. 3-1-3-1

5) The clearing of Y and Z axes are the same as above.

### 3.2 Program display



Press **PROGRAM** key to enter program display that have 5 modes: **【◆PRG】** , **【MDI】** , **【CUR/MOD】** , **【CUR/NXT】** and **【DIR】** . They can be viewed and modified by corresponding soft keys. 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).

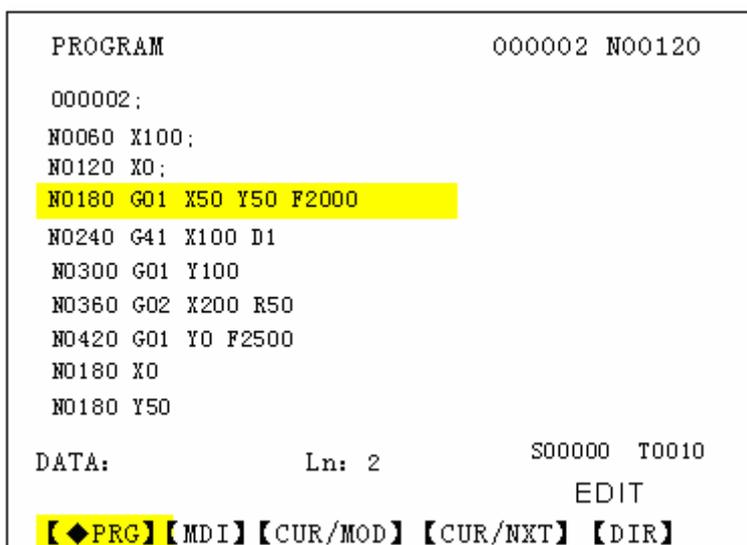


Fig. 3-2-1

Press soft key 【◆PRG】 again, it enters the program EDIT and modification page (see Fig.3-2-2)

```

PROGRAM                                000002 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 T0010
EDIT
【B. EDIT】【B. LOGIN】【CHECK】【SAVE】【RETURN】▶
    
```

Fig. 3-2-2

Press 【▶】 key to enter next page

```

◀【REPLACE】【CUT】【COPY】【PASTE】【RETURN】▶
    
```

Press 【▶】 key to enter next page

```

◀ 【RSTR】 【RETURN】
    
```

Press 【◀】 key to return to last page

**Note** The 【CHECK】 function can only be performed in Auto mode.

【B.EDIT】 and 【B.LOGIN】 are used only in AUTO and DNC mode(background edit function). Functions of 【B.EDIT】 are the same as the <EDIT> mode that is described in CHAPTER 10 “Program Edit”. Save the editing by 【B.LOGIN】 or exit the background EDIT page by 【RETURN】 after editing.

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. 3-2-3)

**Operation**

```

PROGRAM (MDI)                                000002 N00120

000000;
%

-----
( ABSOLUTE )    G00 G17 G90 G94
                  G54 G21 G40 G49
X   0.0000      G98 G15 G50 G69
Y   0.0000      F    0 AF    0
Z   0.0000      S    0 M    30
                  H  0000 D  0000

DATA            Ln: 1                        S00000 T0010
                                           EDIT
【◆PRG】【MDI】【CUR/MOD】【CUR/NXT】【DIR】
    
```

**Fig. 3-2-3**

**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. 3-2-4).

```

PROGRAM (CURRENT/MODAL)                      000002 N00120

(CURRENT)                                     (MODAL)
X           G00      F    1000
Y           G17      S    1000
Z           G90      M    30
A           G94      T    0000
B           G54      H    0000
R           G21      D    0000
I           G40
J           G49      (ABSOLUTE)
K           G11      X    0.0000
P           G98      Y    0.0000
Q           G15      Z    0.0000
F           G50
L           S        G69      SPRM  02500
M           T        G64      SMAX  100000
H           D        G97
DATA            S00000 T0010
                                           MDI
【◆PRG】【MDI】【CUR/MOD】【CUR/NXT】【DIR】
    
```

**Fig. 3-2-4**

**4) Program (CUR/NXT) display**

Press **【CUR/NXT】** soft key to enter current/next interface, It displays the instructions of the blocks being executed and the blocks to be executed. (See Fig. 3-2-5).

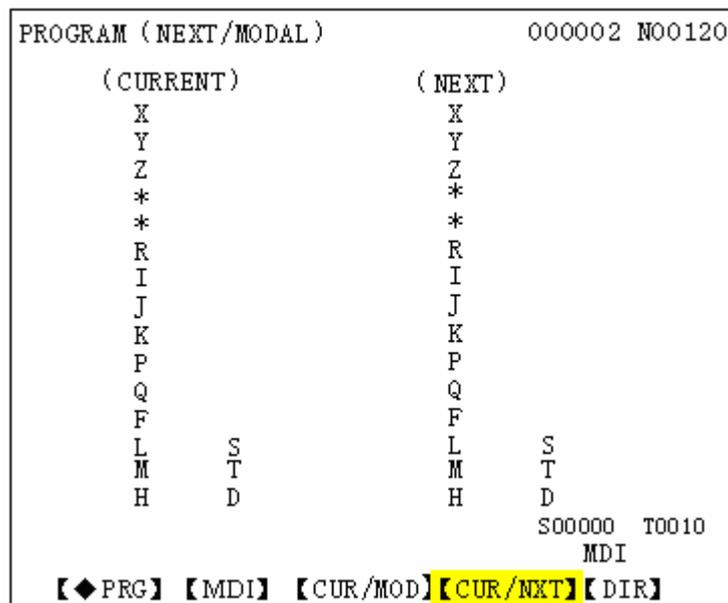


Fig. 3-2-5

5) Program (DIR) display

Press **【DIR】** soft key to enter directory interface, it displays(Fig.3-2-6):

(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.

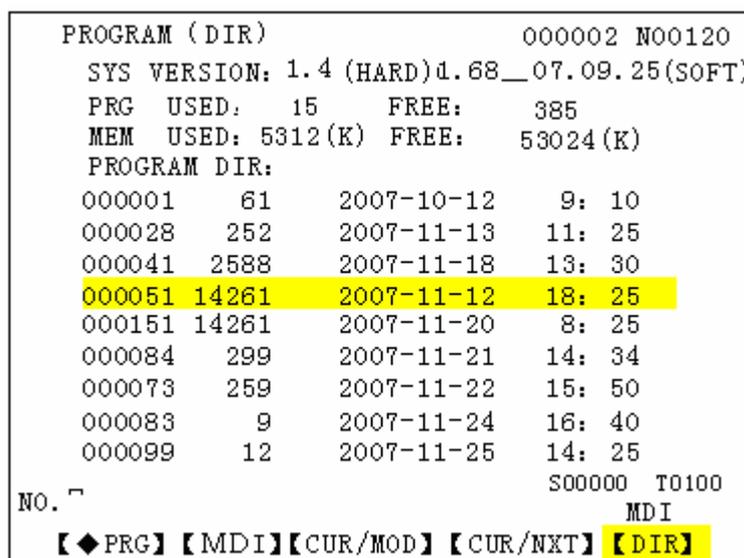
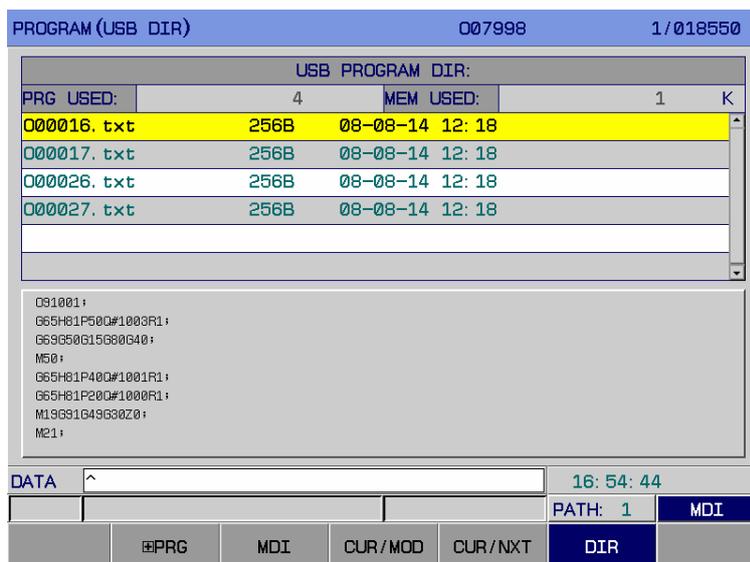


Fig. 3-2-6

II、 Press [DIR] soft key to enter program (USB directory) display page. See Fig.3-2-7:



**Fig. 3-2-7**

**Explanation:** The program numbers in memory can be displayed by the page keys.

### 3.3 System display



Press **系统 SYS** key to enter system page, There are **【OFFSET】**, **【PARAMETER】**, **【MACRO】** and **【SCREW】** modes in this page. And they can be viewed and modified by corresponding soft keys, the steps are as following (see Fig.3-6-1):

#### 3.3.1 Parameter display, modify and setting



Press **PARAMETER** key to enter parameter page. There are **【BITPAR】**, **【NUMPAR】**, **【MACRO1】** and **【MACRO2】** modes in this page. And they can be viewed and modified by corresponding soft keys, the steps are as following:

- 1) Bit parameter page Press **【BITPAR】** soft key to enter this page (see Fig. 3-3-1-1):

| BIT PARAMETER |      |     |      |      |      |      |        |       | 000002 | N00120 |
|---------------|------|-----|------|------|------|------|--------|-------|--------|--------|
| NO.           | DATA |     |      |      |      |      |        |       |        |        |
| 0000          | 1    | 0   | 1    | 0    | 1    | 0    | 1      | 0     |        |        |
|               | **   | **  | SEQ  | **   | **   | INI  | ISO    | **    |        |        |
| 0001          | 1    | 1   | 1    | 1    | 1    | 1    | 0      | 1     |        |        |
|               | SJZ  | **  | **   | **   | **   | **   | **     | MIRI  |        |        |
| 0002          | 1    | 1   | 1    | 1    | 1    | 1    | 0      | 0     |        |        |
|               | ND3  | IOP | **   | **   | ASI1 | SB1  | ASIO   | SBO   |        |        |
| 0003          | 1    | 1   | 1    | 1    | 1    | 0    | 0      | 0     |        |        |
|               | **   | **  | DIR5 | DIR4 | DIRZ | DIRY | DIRX   | INM   |        |        |
| 0004          | 1    | 1   | 1    | 1    | 0    | 0    | 0      | 0     |        |        |
|               | IDG  | **  | **   | XIK  | AZR  | SFD  | DLZ    | JAK   |        |        |
| 0005          | 1    | 1   | 0    | 0    | 0    | 0    | 0      | 0     |        |        |
|               | IPR  | **  | **   | **   | **   | **   | ISC    | **    |        |        |
| DATA          |      |     |      |      |      |      | S00000 | T0010 |        |        |
|               |      |     |      |      |      |      |        | MDI   |        |        |

【BITPAR】
【NUMPAR】
【MACRO1】
【MACRO2】

Fig. 3-3-1-1

See details about this parameter in Appendix 1.

2) Number parameter page Press **【NUMPAR】** soft key to enter this page (see Fig. 3-3-1-2):

| NUM PARAMETER |   | 000002 | N00120 |
|---------------|---|--------|--------|
| NO.           | MEANING   | DATA   |        |
| 0000          | I/O channel, select input/output device             | 0      |        |
| 0001          | communication channel 0 baud rate (DNC)             | 38400  |        |
| 0002          | communication channel 1 baud rate                   | 0      |        |
| 0003          | screen protection wait time(minute)                 | 15     |        |
| 0004          | system interpolation period(1, 2, 4, 8 millisecond) | 1      |        |
| 0005          | number of axis control CNC                          | 4      |        |
| 0006          | program axis name of each axis                      | 0      |        |
| 0007          | set axis name in basic coordinate system            | 0      |        |
| 0008          | servo axis number of each axis                      | 0      |        |
| 0009          | STANDBY   | 0      |        |
| 0010          | external workpiece origin point X offset            | 0.0000 |        |
| 0011          | external workpiece origin point Y offset            | 0.0000 |        |
| DATA          |   | S00000 | T0010  |
|               |   |        | EDIT   |

【BITPAR】
【NUMPAR】
【MACRO1】
【MACRO2】

Fig. 3-3-1-2

See details about this parameter in Appendix 1.

3) Macro variable 1 page Press **【MACRO 1】** soft key to enter this page (see Fig. 3-3-1-3):

**Operation**

| COMMON VARIABLES |      | 000002 N00120   |                 |
|------------------|------|-----------------|-----------------|
| NO.              | DATA | NO.             | DATA            |
| 0000             |      | 0012            |                 |
| 0001             |      | 0013            |                 |
| 0002             |      | 0014            |                 |
| 0003             |      | 0015            |                 |
| 0004             |      | 0016            |                 |
| 0005             |      | 0017            |                 |
| 0006             |      | 0018            |                 |
| 0007             |      | 0019            |                 |
| 0008             |      | 0020            |                 |
| 0009             |      | 0021            |                 |
| 0010             |      | 0022            |                 |
| 0011             |      | 0023            |                 |
| ALWAYS NULL      |      | S00000          | T0010           |
| DATA             |      |                 | EDIT            |
|                  |      | <b>【BITPAR】</b> | <b>【NUMPAR】</b> |
|                  |      | <b>【MACRO1】</b> | <b>【MACRO2】</b> |

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

4) Macro variable 2 page Press **【MACRO 2】** soft key to enter this page (see Fig. 3-3-1-4):

| SYSTEM VARIABLES       |      | 000002 N00120   |                 |
|------------------------|------|-----------------|-----------------|
| NO.                    | DATA | NO.             | DATA            |
| 1000                   | 0    | 1012            | 0               |
| 1001                   | 0    | 1013            | 0               |
| 1002                   | 0    | 1014            | 0               |
| 1003                   | 0    | 1015            | 0               |
| 1004                   | 0    | 1016            | 0               |
| 1005                   | 0    | 1017            | 0               |
| 1006                   | 0    | 1018            | 0               |
| 1007                   | 0    | 1019            | 0               |
| 1008                   | 0    | 1020            | 0               |
| 1009                   | 0    | 1021            | 0               |
| 1010                   | 0    | 1022            | 0               |
| 1011                   | 0    | 1023            | 0               |
| INPUT INTERFACE SIGNAL |      | S00000          | T0010           |
| NO.                    |      |                 | EDIT            |
|                        |      | <b>【BITPAR】</b> | <b>【NUMPAR】</b> |
|                        |      | <b>【MACRO1】</b> | <b>【MACRO2】</b> |

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

**3.3.2 Modification and setting of the parameter values**

- 1) Select MDI mode;
- 2) Enter <SET> mode, input the corresponding password in the 2<sup>nd</sup> page **【PSW】** mode of the “SET” , and <RETURN> to **【ON-OFF】** mode to set the parameter switch for ON.



- 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 SEARCH key, input parameter number and press  key for location (step 4 can be ignored).

5) Input the new parameter values by numerical keys;

6) Press  key to enter and display the parameter value;

7) After all parameters are set and entered, set the parameter switch for OFF.

### 3.3.3 Macro variable display, modification and setting

#### 3.3.3.1 Macro variable display

Press  key to enter macro page, there are **【USER VARIABLE】**, **【SYS VARIABLE】**, sub-modes in this page. They can be viewed or modified by corresponding soft keys, which is shown as following:

1) **USER VARIABLE** page Press **【USER VARIABLE】** soft key to enter this page (Fig. 3-3-3-1-1):

| COMMON VARIABLES |      | 007998 |      | 1/018550 |  |
|------------------|------|--------|------|----------|--|
| NO.              | DATA | NO.    | DATA |          |  |
| 0000             |      | 0012   |      |          |  |
| 0001             |      | 0013   |      |          |  |
| 0002             |      | 0014   |      |          |  |
| 0003             |      | 0015   |      |          |  |
| 0004             |      | 0016   |      |          |  |
| 0005             |      | 0017   |      |          |  |
| 0006             |      | 0018   |      |          |  |
| 0007             |      | 0019   |      |          |  |
| 0008             |      | 0020   |      |          |  |
| 0009             |      | 0021   |      |          |  |
| 0010             |      | 0022   |      |          |  |
| 0011             |      | 0023   |      |          |  |

NOTE: ALWAYS NULL

DATA ^

16:56:48

PATH: 1 MDI

CUSTOMER SYSTEM RETURN

Fig. 3-3-3-1-1

2) **SYS VARIABLE** page Press **【SYS VARIABLE】** soft key to enter this page (see Fig. 3-4-1-3):

**Operation**

| SYSTEM VARIABLES |      | 007998 | 1/018550 |
|------------------|------|--------|----------|
| NO.              | DATA | NO.    | DATA     |
| 1000             | 0    | 1012   | 0        |
| 1001             | 0    | 1013   | 0        |
| 1002             | 0    | 1014   | 0        |
| 1003             | 0    | 1015   | 0        |
| 1004             | 0    | 1016   | 0        |
| 1005             | 0    | 1017   | 0        |
| 1006             | 0    | 1018   | 0        |
| 1007             | 0    | 1019   | 0        |
| 1008             | 0    | 1020   | 0        |
| 1009             | 0    | 1021   | 0        |
| 1010             | 0    | 1022   | 0        |
| 1011             | 0    | 1023   | 0        |

NOTE: INPUT INTERFACE SIGNAL

DATA  16: 57: 03

CUSTOMER SYSTEM PATH: 1 MDI RETURN

**Fig. 3-3-3-1-2**

See chapter 4.7.2 macro variable for the details

**3.3.2 Modification and setting of the offset value**

**3.3.2.1 Modification and setting of the offset value**

The steps for tool offset in Tool Offset page are as follows:

- 1) Press  key to enter the Tool Offset page;
- 2) Move the cursor to the offset number for inputting;
  - Step 1: press page keys to display the offset page to be modified, move the cursor by pressing cursor keys to locate the offset number to be modified.

Step 2: press "SEARCH" key, input offset number and press  key for location.

- 3) In <MDI > mode, input the offset value. After pressing  key, the offset amount is computed by system automatically and displayed on LCD.

**Note 1** During the tool offset setting, the new offset value is ineffective till its offset number T code is specified.

- 2 The offset value can be modified during the program execution. If the value is needed to be effective during the program execution, the modification must be completed before the tool offset number is used.

- 3 If length offset is needed to be added the relative coordinate value of Z axis, the offset value should follow Z code, then it will be automatically accumulated.

**Example**

If Z10 is input, the offset value is the one the actual relative coordinate added by 10.

**3.3.2.2 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) Enter the value for every offset point by sequence in <MDI> mode.

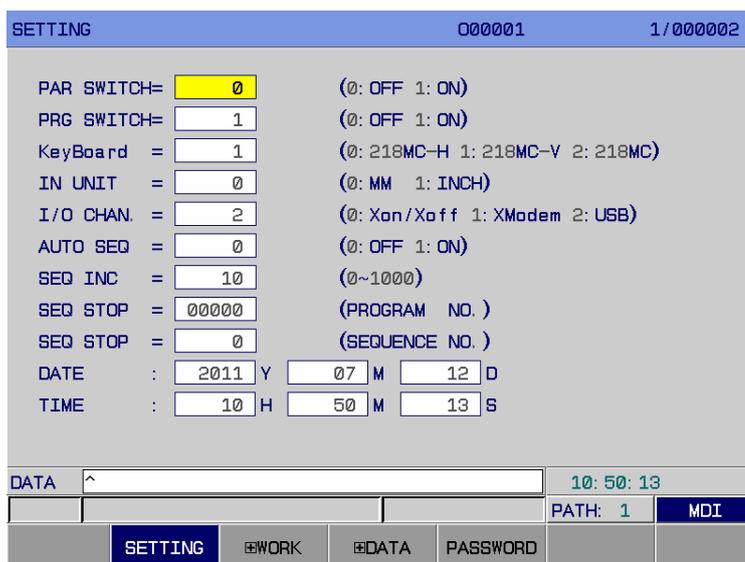
**Note** Refer to Volume Four Connection of *GSK218MC-H Connection and PLC manual* for the pitch setting.

**3.4 Setting display**

**3.4.1 Setting page**

- 1 Entry of page

Pressing  key to enter the SETTING page, there are **【SETTING】** , **【WORKPIECE】** , **【DATA】** , **【PSW】** sub-modes in this page. They can be viewed or modified by corresponding soft keys, the pages are shown as following (see Fig. 3-4-1-1):



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

- 2 **【SETTING】** explanation

Press **【SETTING】** soft key to enter the page shown as Fig.3-4-1-1. After entering the page, the user can view and modify the parameters. The operation steps are as following:

- (a) Enter < MDI> mode;
- (b) Move the cursor to the items to be altered by pressing cursor keys;
- (c) Key in 1 or 0 by following steps:
  - 1) X, Y, Z axis mirror image  
1: Mirror image on    0: Mirror image off
  - 2) ISO code  
When the data in memory are input or output, the code selected:  
1: ISO code                    0: EIA code  
**Note Use ISO code if GSK218MC-H universal programmer is used.**
  - 3) Inch programming  
Set the input unit of the program for inch or mm  
1: inch                    0: mm
  - 4) I/O channel  
To be set by user's requirement.
  - 5) Absolute programming  
0: Absolute programming            1: Incremental programming
  - 6) 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 No.0210.
  - 7) 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.
- (d) Press  key to confirm the entry.

### 3.4.2 Coordinate setting interface

1. Press 【G54-G59】soft key to enter coordinate setting interface, which is shown as following (Fig.3-4-2-1):

| SETTING (G54-G59) |          |  | CUR. COORD. SYS: G54 |          |  | O00001       |          |  | 1/000002 |  |  |
|-------------------|----------|--|----------------------|----------|--|--------------|----------|--|----------|--|--|
| <b>(MACHINE)</b>  |          |  | <b>(G54)</b>         |          |  | <b>(G55)</b> |          |  |          |  |  |
| X                 | 0.000 mm |  | X                    | 0.000 mm |  | X            | 0.000 mm |  |          |  |  |
| Y                 | 0.000 mm |  | Y                    | 0.000 mm |  | Y            | 0.000 mm |  |          |  |  |
| Z                 | 0.000 mm |  | Z                    | 0.000 mm |  | Z            | 0.000 mm |  |          |  |  |
| <b>(EXT)</b>      |          |  | <b>(G56)</b>         |          |  | <b>(G57)</b> |          |  |          |  |  |
| X                 | 0.000 mm |  | X                    | 0.000 mm |  | X            | 0.000 mm |  |          |  |  |
| Y                 | 0.000 mm |  | Y                    | 0.000 mm |  | Y            | 0.000 mm |  |          |  |  |
| Z                 | 0.000 mm |  | Z                    | 0.000 mm |  | Z            | 0.000 mm |  |          |  |  |
| DATA ^            |          |  |                      |          |  | 10:50:34     |          |  |          |  |  |
|                   |          |  |                      |          |  | PATH: 1      |          |  | MDI      |  |  |
| WORK              |          |  | AUTOMEAS             |          |  | +INPUT       |          |  | INPUT    |  |  |
|                   |          |  |                      |          |  | RETURN       |          |  |          |  |  |

Fig. 3-4-2-1

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. 3-4-2-2. 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) |          |  | CUR. COORD. SYS: G54 |          |  | O00001       |          |  | 1/000002 |  |  |
|-------------------|----------|--|----------------------|----------|--|--------------|----------|--|----------|--|--|
| <b>(MACHINE)</b>  |          |  | <b>(G54)</b>         |          |  | <b>(G55)</b> |          |  |          |  |  |
| X                 | 0.000 mm |  | X                    | 0.000 mm |  | X            | 0.000 mm |  |          |  |  |
| Y                 | 0.000 mm |  | Y                    | 0.000 mm |  | Y            | 0.000 mm |  |          |  |  |
| Z                 | 0.000 mm |  | Z                    | 0.000 mm |  | Z            | 0.000 mm |  |          |  |  |
| <b>(EXT)</b>      |          |  | <b>(G56)</b>         |          |  | <b>(G57)</b> |          |  |          |  |  |
| X                 | 0.000 mm |  | X                    | 0.000 mm |  | X            | 0.000 mm |  |          |  |  |
| Y                 | 0.000 mm |  | Y                    | 0.000 mm |  | Y            | 0.000 mm |  |          |  |  |
| Z                 | 0.000 mm |  | Z                    | 0.000 mm |  | Z            | 0.000 mm |  |          |  |  |
| DATA ^            |          |  |                      |          |  | 10:50:34     |          |  |          |  |  |
|                   |          |  |                      |          |  | PATH: 1      |          |  | MDI      |  |  |
| WORK              |          |  | AUTOMEAS             |          |  | +INPUT       |          |  | INPUT    |  |  |
|                   |          |  |                      |          |  | RETURN       |          |  |          |  |  |

Fig. 3-4-2-2

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

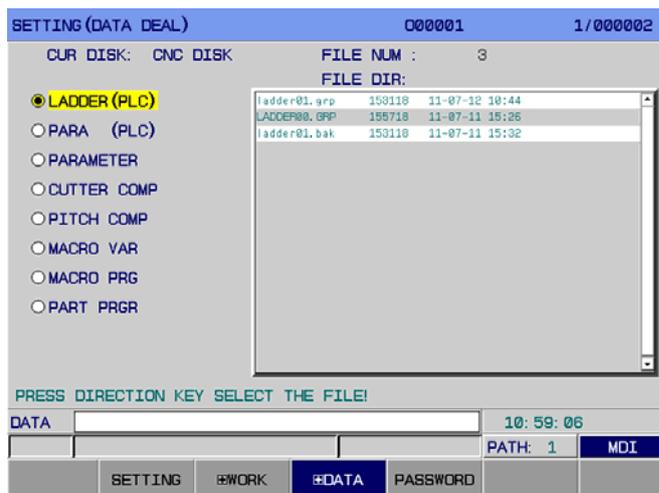


origin, press  key for confirmation.

### 3.4.3 Backup, restore and transfer of the data

In <SETTING> mode the 2<sup>nd</sup> page, press **【DATA】** soft key to enter data page. 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.3-5-6-1)



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

**Operation:**

- 1 In the 2<sup>nd</sup> page of <SETTING> mode, set the corresponding password in **【PSW】** soft key page. 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 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** Data input and output system needs to connect with PC to transfer data by the relevant software.

### 3.4.4 Password authority setting and modification

To prevent the part programs and CNC parameter from malicious modification, the password authority setting is available in this GSK218MC-H 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.3-5-7-1) .

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 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.

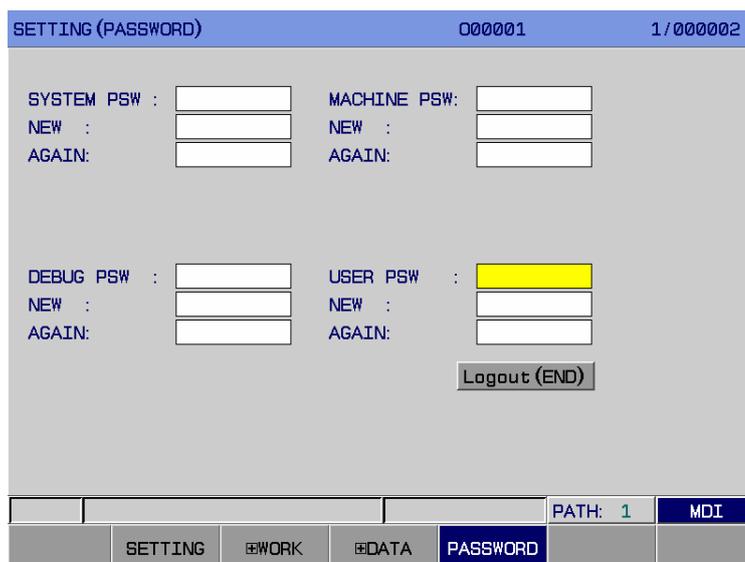


Fig. 3-4-4-1

- 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.

3) Modify the corresponding parameters and setting for the system password;

4) After modification, move the cursor to the "(END)" button by pressing "" key, and it will prompt in the page: " Press ENTER key to confirm the cancellation! "; After pressing



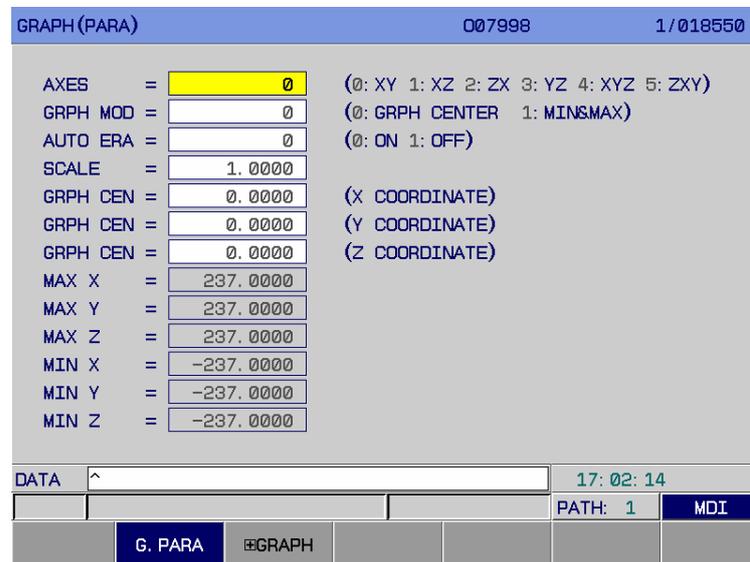
key, it prompts : "Cancellation is Finished! "; press  key, the cursor returns to the password setting item. The password is automatically cancelled when keying in incorrect password or repowering.

### 3.5 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 the corresponding soft keys. (See Fig.3-5-1)



**Fig. 3-5-1**

1) Graphic parameter page Press **【G. PARA】** soft key to enter this page, see Fig.3-5-1.

#### A、 Graphic parameter meaning

- ①Coordinate selection: set drawing plane that has 6 types as shown in the next line
- ②Graphic mode: set graphic display mode
- ③Automatic erase: when it is set to 1, the program graphic is erased automatically in the

next cycle starting after the program is finished.

- ④ Scaling: set drawing ratio
- ⑤ Graphic center: set the coordinate of the LCD center in workpiece coordinate system
- ⑥ 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)

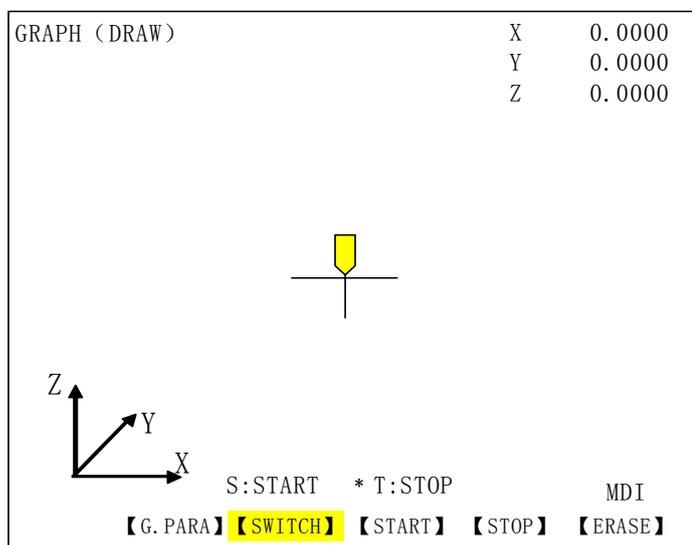
Minimum value of Z axis: the maximum value along X axis in graphics (unit: 0.001mm)

**B、** The graphic parameters setting steps:

- 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 Press **【GRAPH】** soft key to enter this page (See Fig.3-6-2):



**Fig. 3-6-2**

The figure machined can be monitored in graphic page,

- A** Press  key or **【START】** soft key to enter the DRAW START mode, then the sign '\*' is headed to **S: START**;

- B** Press  key or **【STOP】** soft key to enter the DRAW STOP mode, then the sign '\*' is

headed to **T: STOP**;

- C** Press **【SWITCH】** soft key once to switch over the graph in the corresponding **0~5** coordinate display page;



- D** Press  key or **【ERASE】** soft key to erase the graph drawn.

### 3.6 Diagnosis display

The status of DI/DO signals between CNC and the machine, the signals transferred between CNC and PMC, PMC internal data and CNC internal status etc. are shown in the diagnosis display. Refer to *GSK218MC-H CNC System Connection and PMC 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.

#### 3.6.1 Diagnosis data display



Press  key to enter the Diagnose page, which has 5 modes: **【CNC】** , **【PMC→CNC】** , **【MT】** , **【PMC→MT】** and **【WAVE】** . They can also be viewed by pressing the soft keys (See Fig.3-6-1-1 to Fig.3-6-1-5) .

1 NC interface Press **【CNC】** soft key in <DIAGNOSE> page to enter this interface, as is shown in Fig.3-6-1-1:

| DIAGNOSE (NC->PLC) |      |   |   | 007998 |   |   |   | 1/018550 |      |      |   |   |   |   |   |   |   |
|--------------------|------|---|---|--------|---|---|---|----------|------|------|---|---|---|---|---|---|---|
| NO.                | DATA |   |   |        |   |   |   |          | NO.  | DATA |   |   |   |   |   |   |   |
| F000               | 0    | 1 | 0 | 0      | 0 | 0 | 0 | 0        | F012 | 0    | 0 | 0 | 0 | 0 | 0 | 0 | 0 |
| F001               | 0    | 0 | 0 | 0      | 1 | 0 | 0 | 0        | F013 | 0    | 0 | 0 | 0 | 0 | 0 | 0 | 0 |
| F002               | 0    | 0 | 0 | 0      | 0 | 0 | 0 | 0        | F014 | 0    | 0 | 0 | 0 | 0 | 0 | 0 | 0 |
| F003               | 0    | 0 | 0 | 0      | 1 | 0 | 0 | 0        | F015 | 0    | 0 | 0 | 0 | 0 | 0 | 0 | 0 |
| F004               | 0    | 0 | 0 | 0      | 0 | 0 | 0 | 0        | F016 | 0    | 0 | 0 | 0 | 1 | 0 | 0 | 0 |
| F005               | 0    | 0 | 0 | 0      | 0 | 0 | 1 | 1        | F017 | 0    | 0 | 0 | 0 | 0 | 0 | 0 | 0 |
| F006               | 0    | 0 | 0 | 0      | 0 | 0 | 0 | 0        | F018 | 0    | 0 | 0 | 0 | 0 | 1 | 1 | 0 |
| F007               | 0    | 0 | 0 | 0      | 0 | 0 | 0 | 0        | F019 | 0    | 0 | 0 | 0 | 0 | 1 | 1 | 0 |
| F008               | 0    | 0 | 0 | 0      | 0 | 0 | 0 | 0        | F020 | 0    | 0 | 0 | 0 | 0 | 0 | 0 | 0 |
| F009               | 0    | 0 | 0 | 0      | 0 | 0 | 0 | 0        | F021 | 0    | 0 | 0 | 0 | 0 | 0 | 0 | 0 |
| F010               | 0    | 0 | 0 | 0      | 0 | 0 | 0 | 0        | F022 | 0    | 0 | 0 | 0 | 0 | 0 | 0 | 0 |
| F011               | 0    | 0 | 0 | 0      | 0 | 0 | 0 | 0        | F023 | 0    | 0 | 0 | 0 | 0 | 0 | 0 | 0 |

DATA ^  17:03:54

PATH: 1 MDI

F SIGNAL G SIGNAL X SIGNAL Y SIGNAL WAVE

**Fig.3-6-1-1**

This is the signal sent to PLC by CNC system. See *GSK218MC-H CNC System Connection and*

PLC Manual for the meaning and setting of the corresponding diagnosis number.

2 PMC→CNC interface In <DIAGNOSE> page, press【PMC→CNC】soft key to enter PMC→CNC interface, as is shown in Fig.3-6-1-2:

| DIAGNOSE (PLC→NC) |      |   |   | O07998 |   |   |   | 1/018550 |      |      |   |   |   |   |   |   |   |
|-------------------|------|---|---|--------|---|---|---|----------|------|------|---|---|---|---|---|---|---|
| NO.               | DATA |   |   |        |   |   |   |          | NO.  | DATA |   |   |   |   |   |   |   |
| G000              | 0    | 0 | 1 | 1      | 0 | 0 | 1 | 1        | G012 | 0    | 0 | 0 | 0 | 0 | 0 | 0 | 0 |
| G001              | 0    | 0 | 0 | 0      | 0 | 0 | 1 | 0        | G013 | 0    | 0 | 0 | 0 | 0 | 0 | 0 | 0 |
| G002              | 0    | 0 | 0 | 0      | 0 | 0 | 0 | 1        | G014 | 0    | 0 | 0 | 0 | 0 | 0 | 0 | 0 |
| G003              | 0    | 0 | 0 | 0      | 0 | 0 | 0 | 0        | G015 | 0    | 0 | 0 | 0 | 0 | 0 | 0 | 0 |
| G004              | 0    | 0 | 0 | 0      | 0 | 0 | 0 | 0        | G016 | 0    | 1 | 0 | 0 | 0 | 0 | 0 | 0 |
| G005              | 0    | 0 | 0 | 0      | 0 | 0 | 0 | 0        | G017 | 0    | 0 | 0 | 0 | 0 | 1 | 1 | 1 |
| G006              | 0    | 0 | 0 | 0      | 0 | 0 | 0 | 0        | G018 | 0    | 0 | 0 | 0 | 0 | 0 | 0 | 0 |
| G007              | 0    | 0 | 0 | 0      | 0 | 0 | 0 | 0        | G019 | 0    | 0 | 0 | 0 | 0 | 1 | 0 | 1 |
| G008              | 0    | 0 | 0 | 0      | 0 | 0 | 0 | 0        | G020 | 0    | 0 | 0 | 0 | 0 | 1 | 0 | 0 |
| G009              | 0    | 0 | 0 | 0      | 0 | 0 | 0 | 0        | G021 | 0    | 0 | 0 | 0 | 0 | 0 | 0 | 0 |
| G010              | 0    | 0 | 0 | 0      | 0 | 0 | 0 | 0        | G022 | 0    | 0 | 0 | 0 | 0 | 0 | 1 | 0 |
| G011              | 0    | 1 | 0 | 1      | 0 | 0 | 0 | 0        | G023 | 0    | 0 | 0 | 0 | 0 | 0 | 0 | 0 |

DATA ^ 17:04:05

PATH: 1 MDI

F SIGNAL G SIGNAL X SIGNAL Y SIGNAL WAVE

Fig.3-6-1-2

This is the signal sent to CNC system by PMC. See *GSK218MC-H CNC System Connection and PMC Manual* for the meaning and setting of the corresponding diagnosis number.

3 MT In <DIAGNOSE> page, press 【MT】 soft key to enter MT page, as is shown in Fig.3-6-1-3:

| DIAGNOSE (MT→PLC) |      |   |   | O07998 |   |   |   | 1/018550 |      |      |   |   |   |   |   |   |   |
|-------------------|------|---|---|--------|---|---|---|----------|------|------|---|---|---|---|---|---|---|
| NO.               | DATA |   |   |        |   |   |   |          | NO.  | DATA |   |   |   |   |   |   |   |
| X000              | 1    | 1 | 1 | 1      | 1 | 1 | 1 | 1        | X012 | 0    | 0 | 0 | 0 | 0 | 0 | 0 | 0 |
| X001              | 0    | 0 | 0 | 0      | 1 | 0 | 0 | 0        | X013 | 0    | 0 | 0 | 0 | 0 | 0 | 0 | 0 |
| X002              | 1    | 1 | 0 | 1      | 1 | 0 | 1 | 0        | X014 | 0    | 0 | 0 | 0 | 0 | 0 | 0 | 0 |
| X003              | 0    | 0 | 0 | 0      | 0 | 0 | 1 | 1        | X015 | 0    | 0 | 0 | 0 | 0 | 0 | 0 | 0 |
| X004              | 0    | 0 | 0 | 0      | 0 | 0 | 0 | 1        | X016 | 0    | 0 | 0 | 0 | 0 | 0 | 0 | 0 |
| X005              | 0    | 1 | 1 | 1      | 0 | 1 | 0 | 0        | X017 | 0    | 0 | 0 | 0 | 0 | 0 | 0 | 0 |
| X006              | 0    | 0 | 0 | 0      | 0 | 0 | 0 | 0        | X018 | 0    | 0 | 0 | 0 | 0 | 0 | 0 | 0 |
| X007              | 0    | 0 | 0 | 0      | 0 | 1 | 1 | 1        | X019 | 0    | 0 | 0 | 0 | 0 | 0 | 0 | 0 |
| X008              | 0    | 0 | 0 | 0      | 0 | 0 | 0 | 0        | X020 | 0    | 0 | 0 | 0 | 0 | 0 | 0 | 0 |
| X009              | 0    | 0 | 0 | 0      | 0 | 0 | 0 | 0        | X021 | 0    | 0 | 0 | 0 | 0 | 0 | 0 | 0 |
| X010              | 0    | 0 | 0 | 0      | 0 | 0 | 0 | 0        | X022 | 0    | 0 | 0 | 0 | 0 | 0 | 0 | 0 |
| X011              | 0    | 0 | 0 | 0      | 0 | 0 | 0 | 0        | X023 | 0    | 0 | 1 | 0 | 0 | 0 | 0 | 0 |

DATA ^ 17:04:18

PATH: 1 MDI

F SIGNAL G SIGNAL X SIGNAL Y SIGNAL WAVE

Fig.3-6-1-3

This is the signal sent to PMC by machine. See *GSK218MC-H CNC System Connection and PMC Manual* for the meaning and setting of the corresponding diagnosis number.

4 PMC→MT interface In <DIAGNOSE> page, press 【PMC→MT】 soft key to enter PMC→MT interface, as is shown in Fig.3-6-1-4:

**Operation**

| DIAGNOSE (PLC->MT) |                 | O07998 |                 | 1/018550 |  |
|--------------------|-----------------|--------|-----------------|----------|--|
| NO.                | DATA            | NO.    | DATA            |          |  |
| Y000               | 1 0 0 0 0 0 0 1 | Y012   | 0 0 0 0 0 1 0 0 |          |  |
| Y001               | 0 0 0 1 0 0 0 0 | Y013   | 0 0 0 0 0 1 0 0 |          |  |
| Y002               | 0 0 0 0 0 0 0 0 | Y014   | 0 0 0 0 0 0 0 0 |          |  |
| Y003               | 0 0 0 0 0 0 0 0 | Y015   | 0 1 0 0 0 0 0 0 |          |  |
| Y004               | 0 0 0 0 0 0 0 0 | Y016   | 0 0 0 0 0 0 0 0 |          |  |
| Y005               | 0 0 0 0 0 0 0 0 | Y017   | 0 0 0 0 0 0 0 0 |          |  |
| Y006               | 0 0 0 0 0 1 0 0 | Y018   | 0 0 0 0 0 0 0 0 |          |  |
| Y007               | 0 0 0 0 0 0 0 0 | Y019   | 0 0 0 0 0 0 0 0 |          |  |
| Y008               | 0 0 0 0 0 0 0 0 | Y020   | 0 0 0 0 0 0 0 0 |          |  |
| Y009               | 0 0 0 0 0 0 0 0 | Y021   | 0 0 0 0 0 0 0 1 |          |  |
| Y010               | 0 0 0 0 0 0 0 0 | Y022   | 0 0 0 0 0 0 1 1 |          |  |
| Y011               | 0 0 0 0 0 0 0 0 | Y023   | 0 0 0 0 0 0 0 0 |          |  |

DATA ^ [ ] 17:04:29

PATH: 1 MDI

F SIGNAL G SIGNAL X SIGNAL Y SIGNAL WAVE

**Fig.3-6-1-4**

This is the signal sent to machine by PMC. See *GSK218MC-H CNC System Connection and PMC* manual for the meaning and setting of the corresponding diagnosis number.

5 WAVE interface In <DIAGNOSE> page, press **WAVE** soft key to enter WAVE interface, as is shown in Fig.3-6-1-5:

| DIAGNOSE WAVE |   | O07998                              |     | 1/018550 |  |
|---------------|---|-------------------------------------|-----|----------|--|
| AXIS          | 0 | (0: ALL 1: X 2: Y 3: Z 4: A 5: B)   |     |          |  |
| WAVE TYPE     | 0 | (0: Speed 1: Acc 2: Acc Acc)        |     |          |  |
| HOR SCALE     | 5 | VER SCALE                           | 500 |          |  |
|               |   |                                     |     |          |  |
| DATA ^ [ ]    |   | 17:04:46                            |     |          |  |
|               |   | PATH: 1                             |     | MDI      |  |
|               |   | F SIGNAL G SIGNAL X SIGNAL Y SIGNAL |     | WAVE     |  |

**Fig. 3-6-1-5**

AXIS: select the axis name for WAVE

WAVE TYPE: select the WAVE type

HOR, VER SCALE: select the WAVE ratio

Data: in MDI mode, move the cursor to select the data to be modified, and press  key for confirmation.

**3.6.2 Signal viewing**

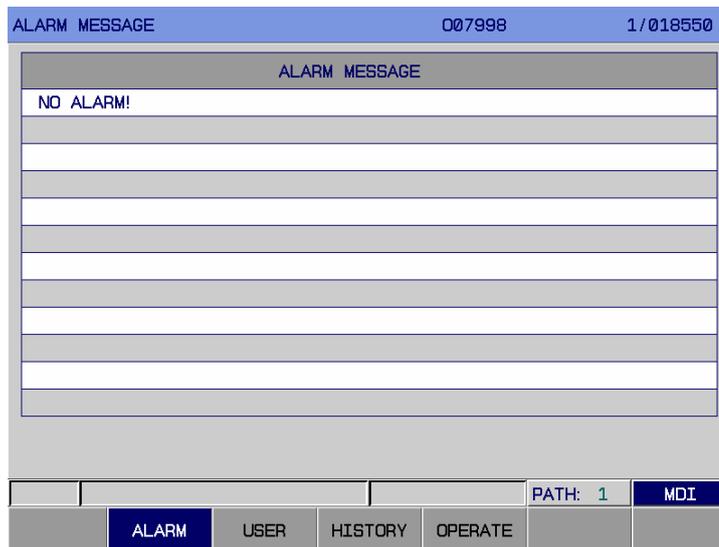
- 1) Press  key to select the DIAGNOSE page.

- 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.
- 4) In **【WAVE】** interface, the feedrate, acceleration, acceleration of acceleration of each axis can be displayed. It is easy to debug the system and find the optimum suited parameters for the drive and the motor.

### 3.7 Alarm display

When an alarm is issued, "ALARM" is displayed at the bottom line of the LCD. Press the  key to display the alarm page, there are 4 modes **【ALARM】**, **【USER】**, **【HISTORY】**, **【OPERATE】** in this page, which can be viewed by the corresponding soft keys (See Fig.3-7-1 to Fig.3-7-4) . They can also be set by bit parameter No: 24#6 for switching to alarm interface if an alarm is issued.

1 Alarm interface In <ALARM> page, press **【ALARM】** soft key to enter this interface, as is shown in Fig.3-7-1:



**Fig.3-7-1**

In alarm page, it displays the message of current P/S alarm number. See details for the alarm in Appendix 2.

2 USER interface In <ALARM> page, press **【USER】** soft key to enter this interface, as is shown in Fig.3-7-2:

**Operation**



**Fig.3-7-2**

See *GSK218MC-H CNC System Connection and PMC* manual for the details of the user alarm.

**Note** The external alarm number can be set and edited by user according to the site conditions. The alarm after editing is input into the system via a transfer software. However, the name of the file edited must be“LadEng01.txt”.

3 HISTORY interface In <ALARM> page, press **【HISTORY】** soft key to enter this interface, as is shown in Fig.3-7-3:



**Fig.3-7-3**

4 OPERATE interface In <ALARM> page, press **【OPERATE】** soft key to enter this interface, as is shown in Fig.3-7-4:

The OPERATE page displays the modification message on the system parameters and ladders, e.g. the modification content and time.



**Operation**



**Fig.3-8-2**

| PLCPARA |      |      |      |      |      |      |      |      | RUN |
|---------|------|------|------|------|------|------|------|------|-----|
| ADDR    | Bit7 | Bit6 | Bit5 | Bit4 | Bit3 | Bit2 | Bit1 | Bit0 |     |
| K000    | 0    | 0    | 0    | 0    | 0    | 0    | 0    | 0    |     |
| K001    | 0    | 0    | 0    | 0    | 0    | 1    | 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    | 1    | 0    |     |
| K006    | 0    | 0    | 0    | 0    | 0    | 0    | 0    | 0    |     |
| K007    | 0    | 0    | 0    | 0    | 0    | 0    | 0    | 0    |     |
| K008    | 0    | 1    | 0    | 0    | 0    | 1    | 0    | 1    |     |
| K009    | 0    | 0    | 0    | 0    | 0    | 0    | 0    | 0    |     |
| K010    | 0    | 0    | 0    | 0    | 0    | 0    | 0    | 0    |     |
| K011    | 0    | 0    | 0    | 0    | 0    | 0    | 0    | 0    |     |

The screenshot also shows a 'DATA' field, a time display of 17:07:34, and a navigation bar with buttons for INFO, PLCGRA, PLCPAR, PLCDGN, and PLCTRA.

**Fig.3-8-3**

| PLCDGN |      |      |      |      |      |      |      |      | RUN |
|--------|------|------|------|------|------|------|------|------|-----|
| ADDR   | Bit7 | Bit6 | Bit5 | Bit4 | Bit3 | Bit2 | Bit1 | Bit0 |     |
| F000   | 0    | 1    | 0    | 0    | 0    | 0    | 0    | 0    |     |
| F001   | 0    | 0    | 0    | 0    | 1    | 0    | 0    | 0    |     |
| F002   | 0    | 0    | 0    | 0    | 0    | 0    | 0    | 0    |     |
| F003   | 0    | 0    | 0    | 0    | 1    | 0    | 0    | 0    |     |
| F004   | 0    | 0    | 0    | 0    | 0    | 0    | 0    | 0    |     |
| F005   | 0    | 0    | 0    | 0    | 0    | 0    | 1    | 1    |     |
| F006   | 0    | 0    | 0    | 0    | 0    | 0    | 0    | 0    |     |
| F007   | 0    | 0    | 0    | 0    | 0    | 0    | 0    | 0    |     |
| F008   | 0    | 0    | 0    | 0    | 0    | 0    | 0    | 0    |     |
| F009   | 0    | 0    | 0    | 0    | 0    | 0    | 0    | 0    |     |
| F010   | 0    | 0    | 0    | 0    | 0    | 0    | 0    | 0    |     |
| F011   | 0    | 0    | 0    | 0    | 0    | 0    | 0    | 0    |     |

The screenshot also shows a 'DATA' field, a time display of 17:07:45, and a navigation bar with buttons for INFO, PLCGRA, PLCPAR, PLCDGN, and PLCTRA.

**Fig.3-8-4**

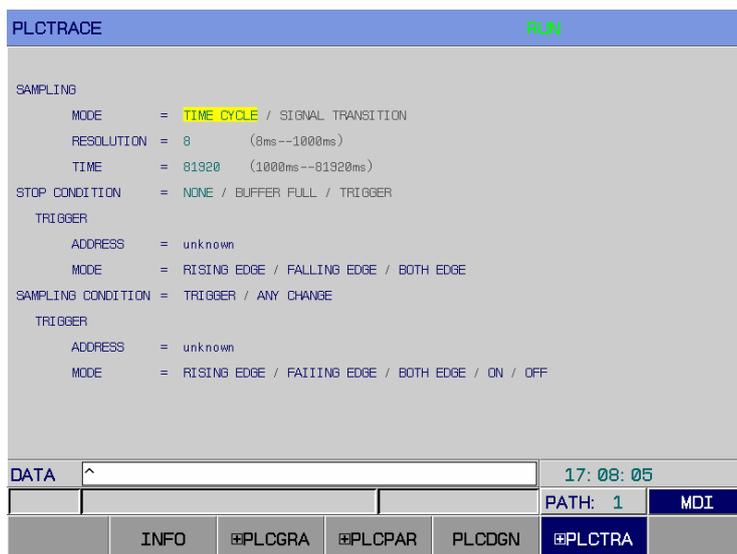


Fig. 3-8-5

**Note** Refer to *GSK218MC-H CNC System Connection and PMC manual* for the PLC ladder modification and relevant message.

### 3.9 Index display



Press the **INDEX** key to display the alarm page, there are 8 modes **【SYS INFO】**, **【OPRT】**, **【ALARM】**, **【G CODE】**, **【PARA】**, **【MACRO】**, **【@PLC.AD】**, **【CALCULA】** in this page, which can be viewed by the corresponding soft keys (See Fig.3-9-1 to Fig.3-9-12)

1 **SYS INFO** interface In <INDEX> page, press **【SYS INFO】** soft key to enter this interface, as is shown in Fig.3-9-1:

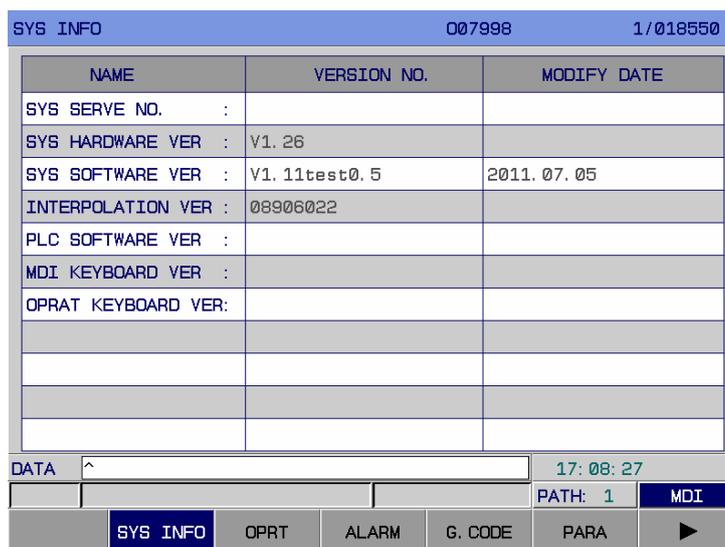
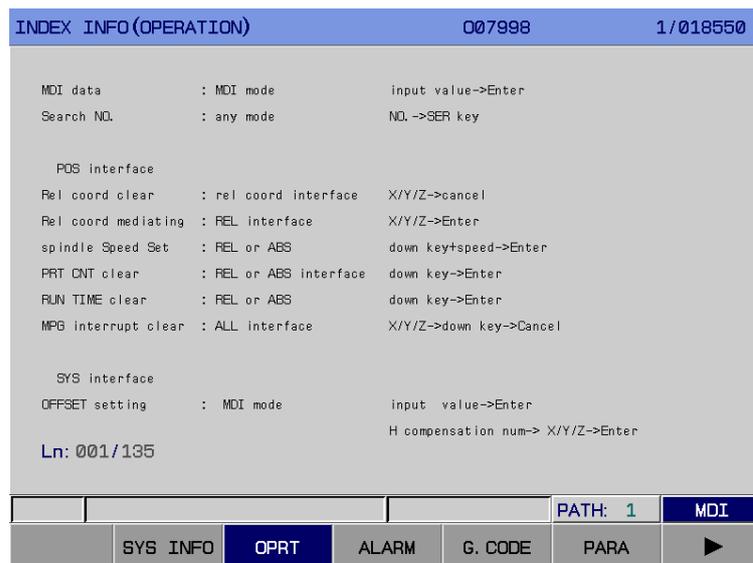


Fig.3-9-1

2 **OPRT** interface In <INDEX> page, press **【OPRT】** soft key to enter this interface, as is

**Operation**

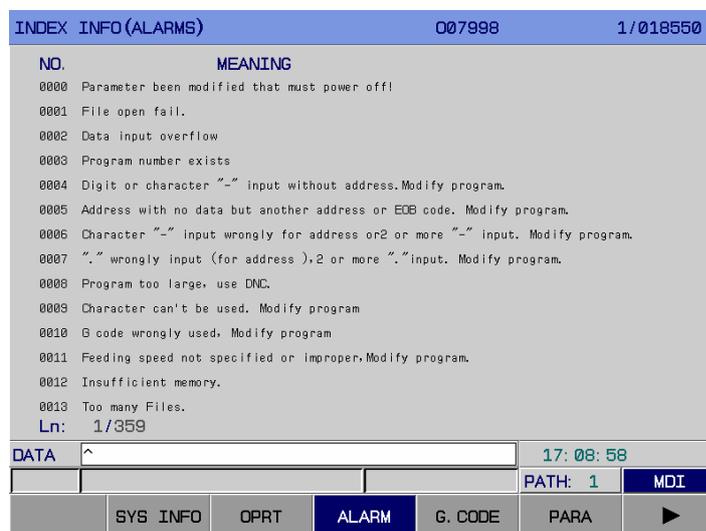
shown in Fig.3-9-2:



**Fig.3-9-2**

The manual operation steps for various interfaces are introduced in <INDEX> page, you may find the corresponding introduction in INDEX pages if you are unfamiliar with some operations.

3 ALARM interface In <INDEX> page, press **【ALARM】** soft key to enter this interface, as is shown in Fig.3-9-3:



**Fig.3-9-3**

In this interface, alarms meaning and operations are shown.

4 G code interface In <INDEX> page, press **【G. CODE】** soft key to enter this interface, as is shown in Fig.3-9-4:



Fig.3-9-4

The meanings of G codes used in system are shown in G code interface, they can be viewed by cursor selection. And the G codes definitions are shown in the down left of the interface, as is shown in Fig.3-9-4. If you want to know the format and usage of a G code, you can press the <ENTER> key on the panel after you select a G code, as is shown in Fig.3-9-5.

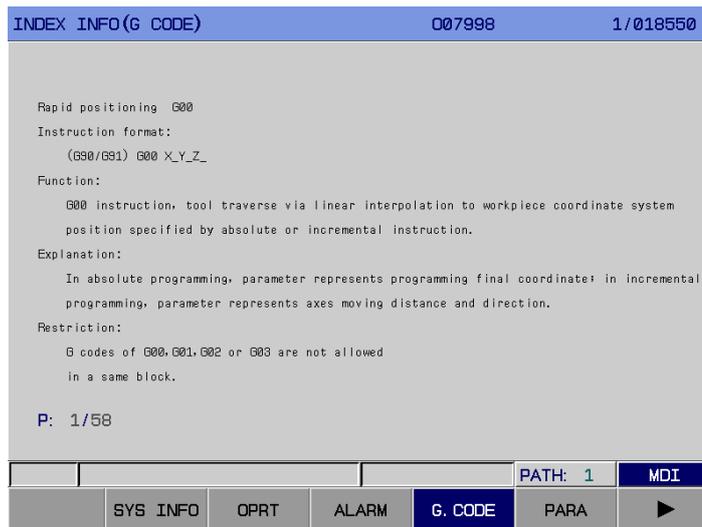


Fig.3-9-5

The format, function, explanation and restriction of instructions are introduced in this page, you may find the corresponding introduction in this page if you are unfamiliar with these instructions.

5 Parameter interface In <INDEX> page, press 【PARA】 soft key to enter this interface, as is shown in Fig.3-9-6:

**Operation**

| INDEX      | INFO (PARAMETER/DIAGNOSE)                   | 007998                                     | 1/018550 |
|------------|---|--|----------|
| <b>NO.</b> | <b>MEANING</b>                              |  |          |
| 0000       | parameters related to "SETTING"             | (bit par.:0000--0002, num par.:0000--0004) |          |
| 0001       | parameters related to axis control          | (bit par.:0003--0008, num par.:0005--0008) |          |
| 0002       | parameters related to coordinate system     | (bit par.:0009--0010, num par.:0010--0065) |          |
| 0003       | parameters related to travel detection      | (bit par.:0011, num par.:0066--0085)       |          |
| 0004       | parameters related to feedrate              | (bit par.:0012--0014, num par.:0086--0104) |          |
| 0005       | parameters related to acc/dec control       | (bit par.:0015--0017, num par.:0105--0157) |          |
| 0006       | parameters related to servo                 | (bit par.:0018, num par.:0160--0186)       |          |
| 0007       | parameters related to backlash              | (bit par.:0018, num par.:0190--0200)       |          |
| 0008       | parameters related to DI/DD                 | (bit par.:0019--0020, num par.:0200--0206) |          |
| 0009       | parameters related to MDI, display and edit | (bit par.:0021--0029, num par.:0210--0214) |          |
| 0010       | parameters related to pitch error comp.     | (bit par.:0037, num par.:0216--0235)       |          |
| 0011       | parameters related to spindle control       | (bit par.:0038, num par.:0240--0260)       |          |
| 0012       | parameters related to tool compensation     | (bit par.:0039--0041, num par.:0266--0267) |          |
| 0013       | parameters related to fixed(canned) cycle   | (bit par.:0042--0043, num par.:0270--0280) |          |
| P: 1/3     |   |  |          |
| DATA       | ^   | 17: 12: 32                                 |          |
|            |   | PATH: 1                                    | MDI      |
| SYS INFO   |   | OPRT                                       | ALARM    |
| G. CODE    |   | PARA                                       | ▶        |

**Fig.3-9-6**

The function parameter settings are introduced in this page, you may find the corresponding introduction in it if you are unfamiliar with some parameter settings.

6 MACRO interface In <INDEX> page, press **【MACRO】** soft key to enter this interface, as is shown in Fig.3-9-7:

| INDEX  | INFO (MACROINSTRUCTION)              | 007998  | 1/018550 |
|--------|--------------------------------------|---------|----------|
| 665    | H(M) P(#I) Q(#J) R(#K)               |         |          |
| M      | : 01-99 operation instruction        |         |          |
| #I     | : operation result (var, seq, alarm) |         |          |
| #J     | : operand 1(variable, invariable)    |         |          |
| #K     | : operand 2(variable, invariable)    |         |          |
| H01:   | #I = #J                              |         |          |
| H02:   | #I = #J+#K                           |         |          |
| H03:   | #I = #J-#K                           |         |          |
| H04:   | #I = #J * #K                         |         |          |
| H05:   | #I = #J / #K                         |         |          |
| H11:   | #I = #J or #K                        |         |          |
| H12:   | #I = #J and #K                       |         |          |
| H13:   | #I = #J xor #K                       |         |          |
| H21:   | #I = sqrt(#J)                        |         |          |
| P: 1/4 |                                      |         |          |
|        |                                      | PATH: 1 | MDI      |
| ◀      | MACRO                                | PLC. AD | CALC     |

**Fig.3-9-7**

The MACRO format and operation instructions are introduced in this page, the local variables, common variables and the system setting range are also shown in this page, you may find the corresponding introduction in it if you are unfamiliar with the macro instruction operations.

7 PLCAD interface In <INDEX> page, press **【PLCAD】** soft key to enter this page,. PLC address interface has four address interfaces: **【F ADDRESS】**, **【G ADDRESS】**, **【X ADDRESS】**, **【Y ADDRESS】** as is shown in Fig. 3-9-8~Fig.3-9-11:

| INDEX              | INFO (PLC ADDRESS) | 007998             | 1/018550 |
|--------------------|--------------------|--------------------|----------|
| * SYMBOL / MEANING |                    |                    |          |
| F000#4             | SPL                | FEED HOLD          |          |
| F000#5             | STL                | Cycle start        |          |
| F000#6             | SA                 | Servo ready        |          |
| F000#7             | OP                 | Auto run           |          |
| F001#0             | AL                 | Alarm              |          |
| F001#1             | RST                | Reset              |          |
| F001#3             | SAR                | Spindle rev arrive |          |
| F001#4             | ENB                | Spindle enable     |          |
| F001#5             | TAP                | Tapping            |          |
| F001#6             | DTAP               | Tapping exe.       |          |
| F001#7             | MTAP               | G63 Tapping mode   |          |
| F002#3             | THRD               | Threading          |          |
| F002#4             | SRMIV              | Program start      |          |
| F002#6             | CUT                | Cutting            |          |
| P: 1/16            |                    |                    |          |
|                    |                    |                    | PATH: 1  |
|                    |                    |                    | MDI      |
|                    | F ADDR             | G ADDR             | X ADDR   |
|                    |                    |                    | Y ADDR   |
|                    |                    |                    | RETURN   |

**Fig.3-9-8**

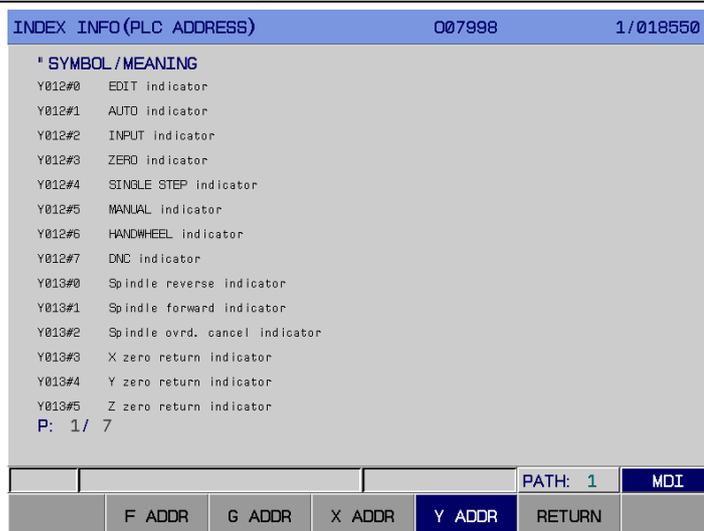
| INDEX              | INFO (PLC ADDRESS) | 007998                                   | 1/018550 |
|--------------------|--------------------|--|----------|
| * SYMBOL / MEANING |                    |  |          |
| G000#0             | FIN                | MST End signal                           |          |
| G000#1             | MFIN               | Miscellaneous function completion signal |          |
| G000#4             | SFIN               | Spindle function completion signal       |          |
| G000#5             | TFIN               | Tool function completion signal          |          |
| G001#0             | ESP                | Emergency stop                           |          |
| G001#1             | SKIPP              | Skip                                     |          |
| G002#0             | GR1                | Gear(input)                              |          |
| G002#1             | GR2                | Gear(input)                              |          |
| G002#2             | GR3                | Gear(input)                              |          |
| G002#4             | GEAR               | Gear in-position(input)                  |          |
| G003#1             | RGTAP              | Rigid tapping                            |          |
| G009#1             | UIINT              | Macroprogram interruption                |          |
| G010#0             | MT1                | Mirror image                             |          |
| G010#1             | MT2                | Mirror image                             |          |
| P: 1/10            |                    |  |          |
|                    |                    |  | PATH: 1  |
|                    |                    |  | MDI      |
|                    | F ADDR             | G ADDR                                   | X ADDR   |
|                    |                    |  | Y ADDR   |
|                    |                    |  | RETURN   |

**Fig.3-9-9**

| INDEX              | INFO (PLC ADDRESS) | 007998 | 1/018550 |
|--------------------|--------------------|--------|----------|
| * SYMBOL / MEANING |                    |        |          |
| X020#0             | MT-EDIT            |        |          |
| X020#1             | MT-AUTO            |        |          |
| X020#2             | MT-INPUT           |        |          |
| X020#3             | MT-ZERO            |        |          |
| X020#4             | MT-SINGLE STEP     |        |          |
| X020#5             | MT-MANUAL          |        |          |
| X020#6             | MT-HANDWHEEL       |        |          |
| X020#7             | MT-DNC             |        |          |
| X021#0             | MT-SKIP            |        |          |
| X021#1             | MT-SINGLE BLOCK    |        |          |
| X021#2             | MT-DRY RUN         |        |          |
| X021#3             | MT-MST LOCK        |        |          |
| X021#4             | MT-MACHINE LOCK    |        |          |
| X021#5             | MT-OPTIONAL HALT   |        |          |
| P: 1/ 6            |                    |        |          |
|                    |                    |        | PATH: 1  |
|                    |                    |        | MDI      |
|                    | F ADDR             | G ADDR | X ADDR   |
|                    |                    |        | Y ADDR   |
|                    |                    |        | RETURN   |

**Fig.3-9-10**

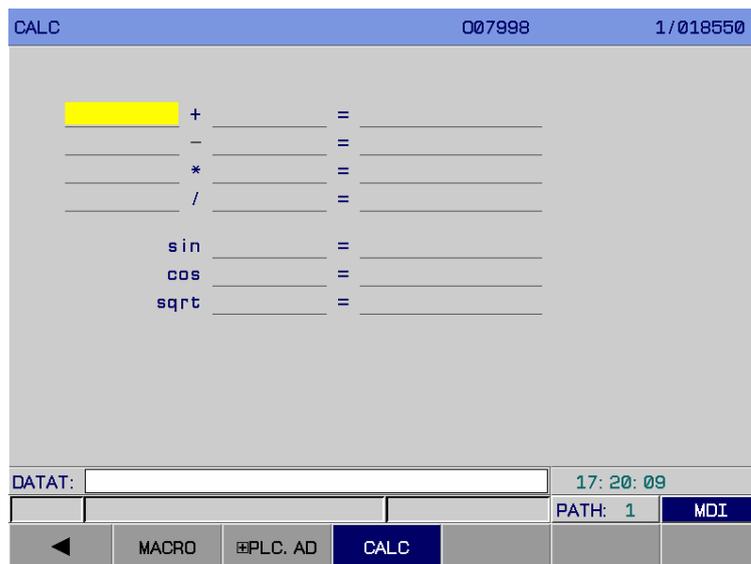
**Operation**



**Fig.3-9-11**

The PLC addresses, signs, meanings are introduced in this page, you may find the corresponding introduction in it if you are unfamiliar with the PLC addresses.

8 CALCULA interface In the 2<sup>nd</sup> page of <INDEX> interface, press **【CALCULA】** soft key to enter this interface, as is shown in Fig.3-9-12:



**Fig.3-9-12**

The operation formats of addition, subtraction, multiplication, division, sine, cosine, extraction are shown in this interface. The cursor may be moved to the space for inputting, and press <ENTER> key for confirmation. After the data input is completed, the system will calculate automatically and input the result to the space behind the “=” sign.

## 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 MANUAL mode, the 3 axes can be moved by MANUAL feed or manual rapid traverse separately.

#### 4.1.1 Manual feed

X axis can be moved to positive or negative direction by pressing and holding the  or  key in Feed Axis and Direction Selection area, the feedrate can be changed by feedrate override, If the key is released, the X axis movement stops, 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 MANUAL 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 P170~173.

**2** The effective manual rapid traverse before reference return is set by the bit parameter No.12.0.

#### 4.1.3 MANUAL feedrate and manual rapid traverse speed selection

The manual feedrate override classified for 20 gears (0%--200%) is available in MANUAL feed by band selector rselection.

|  |  |   |  |
|--|--|---|--|
|  F0 |  25% |  50% |  100% |
| 0.001  | 0.01   | 0.1   | 1  |

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% (25%, 50%, 100% overrides are set by parameters No.088~092, F0 override by number parameter P93).

**Note** The rapid overrides are effective for the following speed:

- (1) G00 rapid traverse
- (2) Rapid traverse in canned cycle
- (3) Rapid traverse in G28
- (4) Manual rapid traverse

Example: If the rapid traverse speed is 6m/min and override is 50%, speed is 3m/min.

**Note** 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 MANUAL mode after a dwell operation, the manual operation is available. Move the axes manually then shift to Auto mode,



press 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:

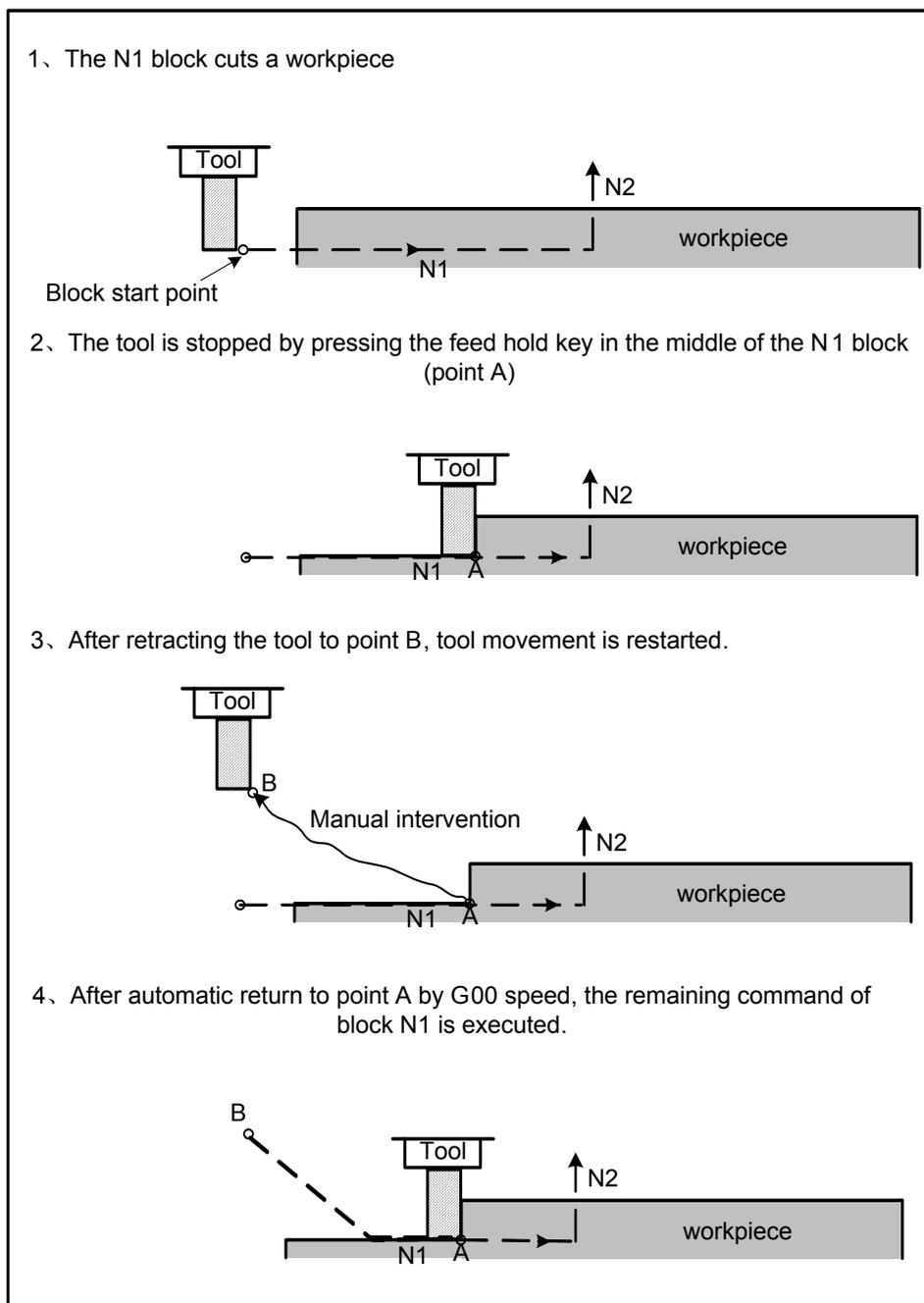


Fig. 4-1-4-1

## 4.2 Spindle control

### 4.2.1 Spindle CCW



: The spindle is started for CCW rotation if this key is pressed in MANUAL./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 gear control for spindle is set by the 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 MANUAL 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.

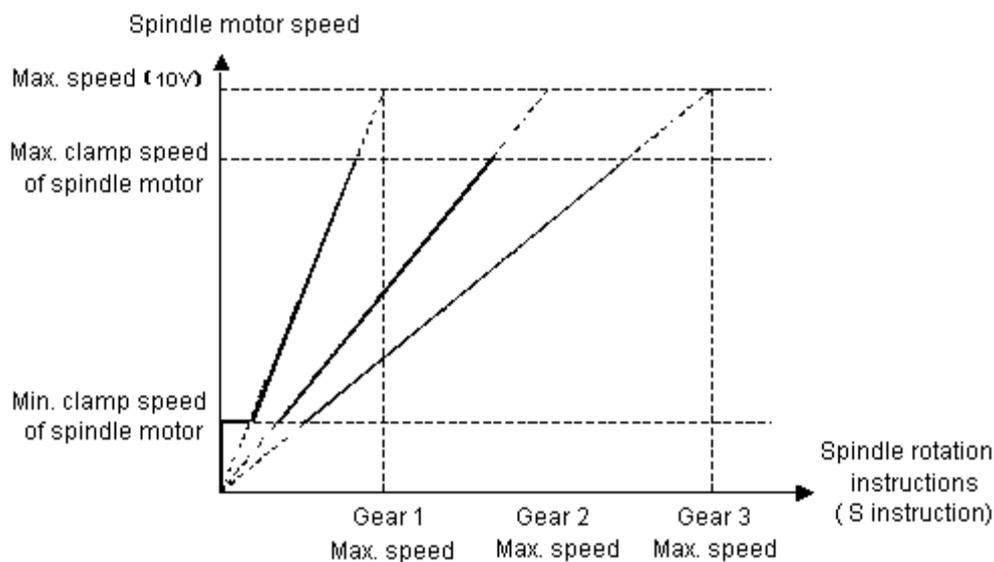


Fig. 4-2-4-1

**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.

### 4.3.3 Chip removal



: Compound key. The chip removal function is switched between ON and OFF by pressing this key. The indicator lighting up is for ON, gone out for OFF.

### 4.3.4 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, while gone out for OFF.

## 5 Step Operation

### 5.1 Step feed



Press **STEP** 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. E.g. If press **0.1** key, in <POSITION> interface it displays a step: 0.100 (See Fig. 5-1-1-1):

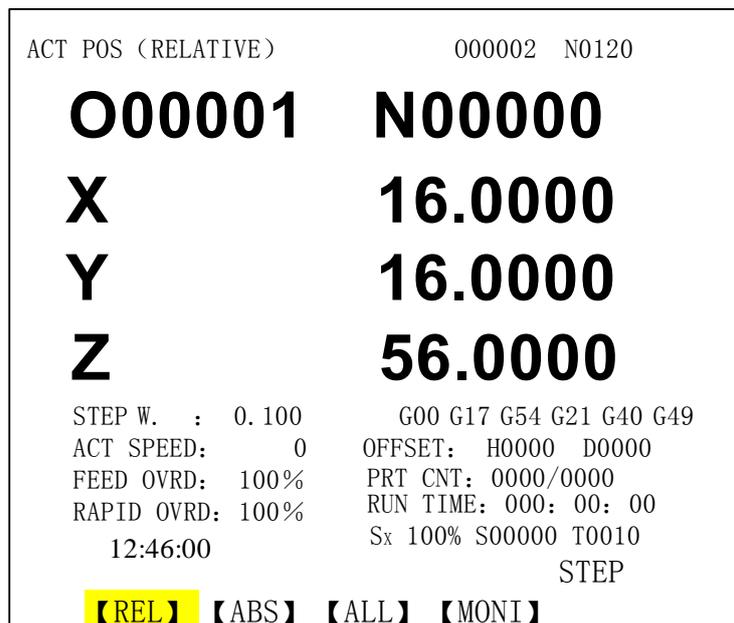


Fig. 5-1-1-1

The machine axis moves 0.1mm when pressing this key once.

#### 5.1.2 Selection of moving axis and direction

X axis may be moved in positive or negative direction by pressing axis and direction key



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

- 1 The step moving speed is identical with the MANUAL feedrate.



- 2 The rapid override is effective after the key is pressed for rapid traverse.

## 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. See details in the Section 6.2 Control in MPG Interruption.

## 5.3 Auxiliary control in Step mode

The auxiliary control in Step mode is the same as that in MANUAL 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. e.g.



If press  key, it displays the MPG increment in <POSITION> interface: 0.100 (See Fig.6-1-1-1) :

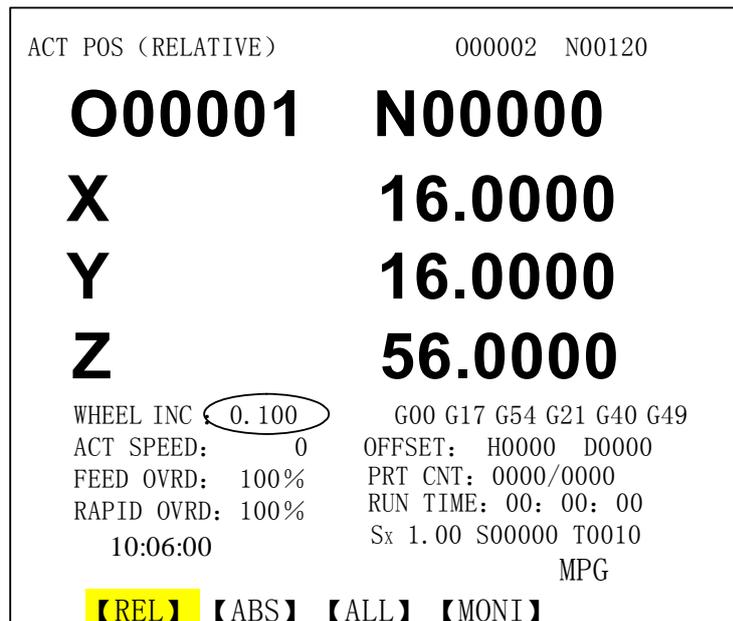


Fig. 6-1-1-1

#### 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) :

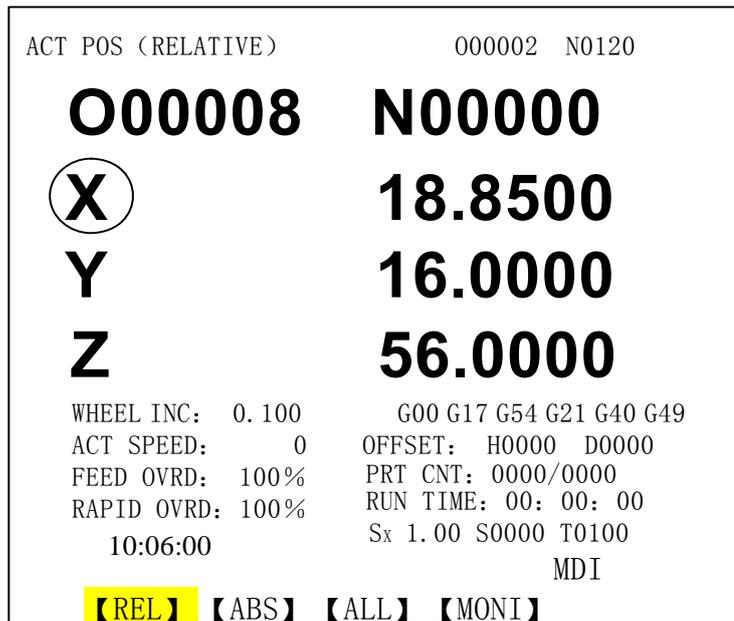


Fig. 6-1-2-1

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, 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

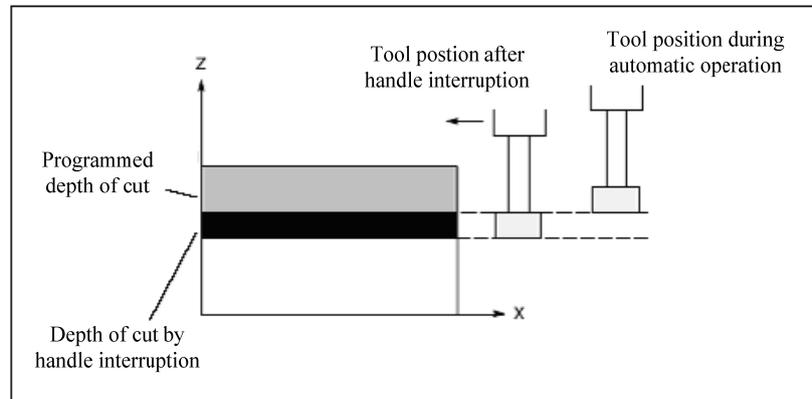
|                                   | Moving amount of a handwheel scale |      |     |   |
|-----------------------------------|------------------------------------|------|-----|---|
| <b>MPG increment (mm)</b>         | 0.001                              | 0.01 | 0.1 | 1 |
| <b>Machine moving amount (mm)</b> | 0.001                              | 0.01 | 0.1 | 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) For the tool offset by handwheel, move Z axis downward or X, Y axis parallel modify the coordinate system.
- 3) 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.

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 |              | 000002 N00120 |
| (RELATIVE)      | (ABSOLUTE)   | (MACHING)     |
| 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 T0010  |
|                 |              | MPG           |
| <b>【REL】</b>    | <b>【ABS】</b> | <b>【ALL】</b>  |
|                 |              | <b>【MONI】</b> |

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

The MPG 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.

### 6.2.2 Relation of MPG interruption with other functions

**Table 6-2-2-1**

| 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 MANUAL mode. See Section 4.2 and 4.3 for details.

## 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.

### 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 OPTIONAL STOP (M01) 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.

#### 5 Press EMERGENCY STOP button

See Section 2.3.2.

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 MANUAL, MPG, Step mode, the machine interruption stops immediately.
- 3) If the control is switched to Machine zero interface, the machine slows down to stop.

## 7.4 Auto running from an arbitrary block

This system permits the current program to be executed from an arbitrary block of it. The steps are as following:

- 1、 Press  key to enter MANUAL mode, start the spindle and other miscellaneous functions.
- 2、 Press  key to enter Edit mode, then press  key to enter program page, select the program to be executed in 【DIR】 ;
- 3、 Open the program and move the cursor to the block to be executed;
- 4、 Start spindle and other miscellaneous functions by pressing  key to enter MANUAL mode;
- 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);**
- 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 (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;
  - 2 The effectiveness of dry run in cutting feed is set by the bit parameter No:12#6.
  - 3 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.



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.
  - 2 The Single Block function is ineffective if the subprogram calling (M98) or the subprogram calling return(M99) instruction is specified. But for a block with M98 or M99, if M98 or M99 block contains an address other than N, O, P, the Single Block function is effective.

## 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 M.S.T. LOCK 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 (5%) till 150% each time the  key is pressed;

The feedrate override descends for a gear (5%) till 0 each time the  key is pressed.

**Note 1** F value in feedrate override program

$$\text{The actual federate} = \text{F value specified} \times \text{feedrate override}$$

During auto running, the rapid traverse speed can be selected by pressing

|       |      |     |   |
|-------|------|-----|---|
|       |      |     |   |
| 0.001 | 0.01 | 0.1 | 1 |

key. The 4 gears override of F0, 25%, 50%, 100% are available for

the rapid traverse.

- The rapid traverse speed value overridden by rapid override and number parameters No.088, No.089, No.090 can be obtained by following equation:

$$\text{The actual rapid traverse speed of X axis} = \text{the value set by parameter No.088} \times \text{rapid override}$$

If the override is F0, the axis stop is set by bit parameter No.12.4. If it is set for non-stop 0, the actual rapid traverse speed is set by number parameter No.093 (for all axes).  
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(5%) till 150% each time the  key is pressed;

The spindle override descends for a gear(5%) 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 No.258, 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 **【◆PRG】** soft key to enter the background edit interface, as is shown in Fig.7-11-1.

**Operation**

```
PROGRAM  
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  
  
AUTO  
【BG. EDT】【BG. END】【CHECK】【SAVE】【RETURN】【▶】
```

**Fig. 7-11-1**

Press **【BG.EDT】** soft key to enter the program background edit interface, the program editing operation is the same as that in Edit mode(Refer to Chapter 10 **Program Edit** in this manual). Then press **【B.LOG】** soft key to save the edited program and exit this interface.

## 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 )                 |       | 000002 N0120 |             |
|---|-------|--------------|-------------|
| ( CURRENT )                               |       | ( MODAL )    |             |
| G0  | X 50  | G00          | F 1000      |
|   | Y 100 | G17          | S 1000      |
|   | Z     | G90          | M 30        |
|   | A     | G94          | T 0000      |
|   | B     | G54          | H 0000      |
|   | R     | G21          | D 0000      |
|   | I     | G40          | (ABSOLUTE)  |
|   | J     | G49          | X 0.0000    |
|   | K     | G11          | Y 0.0000    |
|   | P     | G98          | Z 0.0000    |
|   | Q     | G15          |             |
|   | F     | G50          |             |
|   | L S   | G69          | SPRM 02500  |
|   | M T   | G64          | SMAX 100000 |
|   | H D   | G97          | S0000 T0100 |
| DATA                                      |       |              | MDI         |
| <b>【◆PRG】【MDI】【CUR/MOD】【CUR/NXT】【DIR】</b> |       |              |             |

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.**
  - 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,  key can be pressed to cancel the input word

by word, or 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 MANUAL mode by a dwell, it will execute manual intervention, see section 4.1.4 Manual interruption.

When the control is switched to Step, MPG, MANUAL, Machine Zero mode during the program execution in Auto, MDI, DNC mode, the system will execute deceleration and stop.

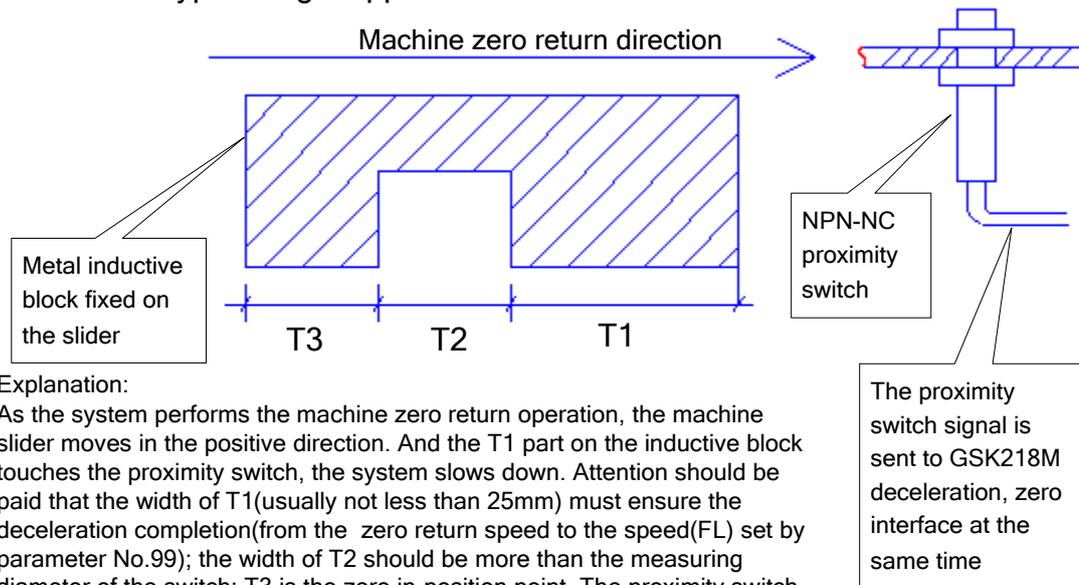
## 9 Machine Zero Operation

### 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.

#### Zero return type using stepper motor with no encoder: A



#### Explanation:

As the system performs the machine zero return operation, the machine slider moves in the positive direction. And the T1 part on the inductive block touches the proximity switch, the system slows down. Attention should be paid that the width of T1 (usually not less than 25mm) must ensure the deceleration completion (from the zero return speed to the speed (FL) set by parameter No.99); the width of T2 should be more than the measuring diameter of the switch; T3 is the zero in-position point. The proximity switch is NPN normal-close type. i.e. The intervals of T1 and T3 are in break-off state. The intervals of T2 and others are through.

#### Machine zero type of GSK218M system- A

Fig. 9-1-1

Zero return type using stepper motor with no encoder: B

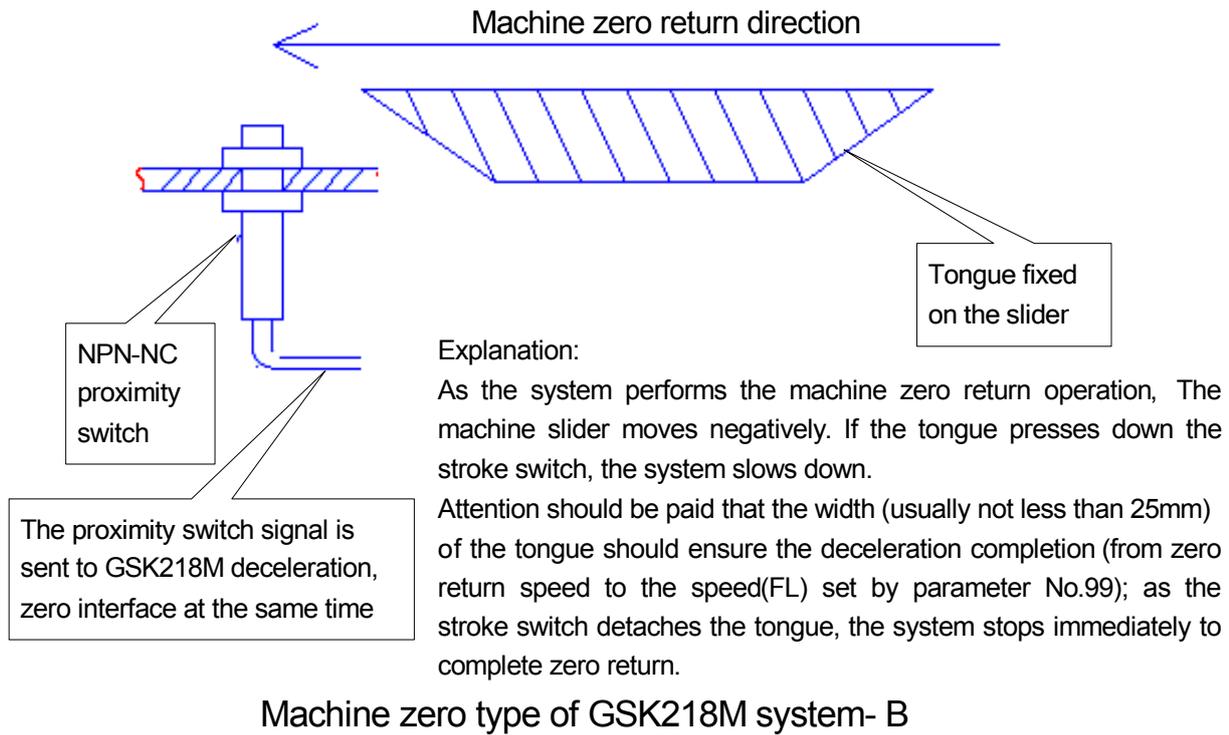


Fig. 9-1-2

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 **No.: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 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.

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.

- 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).

## 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 program page, then press **【◆PROG】** soft key to enter the program edit and modification interface, as is shown in Fig.10-1-1:

```

PROGRAM

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

EDIT

【BG. EDT】【BG. END】【CHECK】【SAVE】【RETURN】【▶】
    
```

Fig.10-1-1

Press **【▶】** soft key to enter next page

```

【◀】【REPLACE】【CUT】 【COPY】 【PASTE】【RETURN】【▶】
    
```

Press **【▶】** soft key to enter next page

```

【◀】【RSTR】 【RETURN】
    
```

Press **【◀】** soft key to return to last page

```

【◀】【REPLACE】【CUT】 【COPY】 【PASTE】【RETURN】【▶】
    
```

The replacement, cut, copy, paste, reset operations etc. can be done by pressing the corresponding soft keys.

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.

**Note** The maximum lines a program file contains are 200,000.

## 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                                000002 N0120

MIRROR X = 1 (0: OFF 1: ON)
MIRROR Y = 1 (0: OFF 1: ON)
MIRROR Z = 1 (0: OFF 1: ON)
CODE = 1 (0: EIA, 1: ISO)
IN UNIT = 0 (0: MM, 1: INCH)
I/O CHAN. = 0 (0—3 CHANNEL NO.)
ABS PRG = 0 (0: ABS, 1: INC)
AUTO SEQ = 1 (0: OFF 1: ON)
SEQ STOP = 0000 (PROGRAM NO.)
SEQ STOP = 0000 (SEQUENCE NO.)

2006 Y 11 M 14 D 14 H 26 M 45 S
DATA                                S0000 T0100
                                EDIT
【SETTING】【SWITCH】【G54-G59】【PANEL】【SERVO】▶
    
```

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 No.0210.

### 10.1.1.2 Program input

1、Press  key to enter Edit mode;

2、Press  key to enter program page (See Fig. 10-1-1-2-1);

```

PROGRAM                                000002 N0180

000003;
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: 3                S0000 T0100
                                EDIT
【◆PRG】 【MDI】 【CUR/MOD】 【CUR/NXT】 【DIR】
    
```

Fig. 10-1-1-2-1

**Operation**

3 Press address key , then key in numerical keys    



by sequence (an example by setting up a program named O00002), it displays O00002 behind the data column (See Fig. 10-1-1-2-2) :

```

PROGRAM                                000002 N0180

000003;
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: 00002                Ln: 3                S0000 T0010
                                           EDIT
【◆PRG】 【MDI】 【CUR/MOD】 【CUR/NXT】 【DIR】
    
```

**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                                000002 N0180

000002;

DATA:                                S0000 T0100
                                           EDIT
【◆PRG】 【MDI】 【CUR/MOD】 【CUR/NXT】 【DIR】
    
```

**Fig. 10-1-1-2-3**

5 Input the blocks programmed word by word, then press interface switching key (e.g



page) or the mode switchover key, the program will be saved automatically and the program input is finished.

**Note 1** In Edit mode, only the complete word can be entered. Single letter and numerical number input is not supported by system.

**2** If word error is found in program inputting, it can be cancelled by pressing



key to delete one by one or pressing



key to delete the whole

word.

### 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 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 Select mode: <Edit >or <Auto>

2 Look up the target program in 【DIR】 page;

3 Press  key to enter the target program;

4 Key in the word or sequence number to be searched and press UP or DOWN 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 sequence number, word searching reaches the end of the program.

**2**、The searching for sequence number、 word and line number may be performed in 【AUTO】 and 【EDIT】 mode, but in 【AUTO】 mode, it can only be performed in the background edit interface.

**10.1.1.4 Location of the cursor**

Select Edit mode, then press  key to display the program.

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 scroll the screen upward, and the cursor moves to the first line and first column of the last page; if it is scrolled to the program beginning, the cursor locates at the second line and the first column of the page.

f) Press  key to scroll the screen downward, and the cursor moves to the first line and first column of the next page; if it is scrolled to the program end, the cursor locates at the last line and the first column of the program.

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. 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.

If the  key is pressed continuously, the words to the right of the cursor 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. 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.

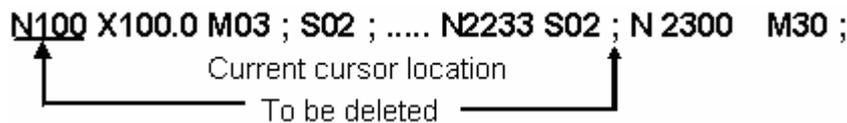


Fig. 10-1-1-7-1

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.10-1-1-7-1 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) Press  key to display the program, there are two ways to delete program;

1. Key in 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.

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.

### 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 page; in **【DIR】** page 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 **【DIR】** page, the name of the new program copied can be viewed.
- The copy of the program can also be done in the program edit page:

- 1 Press address key  and key in the new program number;
- 2 Press the **【COPY】** soft key, the file will be copied and the control enters the new program edit page.
- 3 Return to **【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.

The copy of the program can also be done in the program edit page:

- 1 Locate the cursor to the beginning of the blocks to be copied;
- 2 Key in the last character of the blocks to be copied;
- 3 Press **【COPY】** soft key, the blocks from the cursor to the character keyed in will be copied.
- 4 Locate the cursor to the position to be pasted, press **【PASTE】** soft key to complete the paste.

**Note** If several words to be copied are same in program, it will copy the blocks to the word nearest to the cursor location.

### 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 **【CUT】** soft key , the block will be cut into clip board.
- e) Locate the cursor to the position to be pasted, and press **【PASTE】**soft key, the block will be pasted.

**Note** If several words to be cut are same in program, it will cut the blocks to the word nearest to the cursor location.

### 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 **【REPLACE】** soft key, the character where the cursor locates will be replaced by the new one.



The block replacement can also be done by the  key on the panel, see details in Section 10.1.1.5.

**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) Press address key , key in the new name;



d) Press key to complete the renaming.

### 10.1.9 Program restart

The program restart function is used under the situation that accident occurs during running, such as tool braking-off, system restarting after power-off, emergency stop. After the accident is eliminated, this function can be used for returning to program braking-off position to go on execution and retracting to original point by G00.

The steps for program restart are as following:

1 Solve the machine accident such as tool change, offset changing, machine zero.



2 In <AUTO> mode, press the key on the panel.



3 Press key to enter the program page, then press 【RESTART】 soft key to enter program restart interface (Fig.10-1-9-1)

| PROGRAM RESTART       |     |                   |       | 000002            |         |                        |      | 2/000002 |    |        |  |
|-----------------------|-----|-------------------|-------|-------------------|---------|------------------------|------|----------|----|--------|--|
|                       |     | <b>(DISTANCE)</b> |       | <b>(ABSOLUTE)</b> |         | <b>(REM DIST)</b>      |      |          |    |        |  |
| (1)                   | X   | 61.680            | mm    | X                 | -1.727  | mm                     | X    | 63.407   | mm |        |  |
| (2)                   | Y   | -48.490           | mm    | Y                 | -47.897 | mm                     | Y    | -0.593   | mm |        |  |
| (3)                   | Z   | -6.186            | mm    | Z                 | -5.480  | mm                     | Z    | -0.706   | mm |        |  |
| <b>(LOADED MODAL)</b> |     |                   |       |                   |         | <b>(CURRENT MODAL)</b> |      |          |    |        |  |
| G00                   | G49 | F                 | 300   | G00               | G49     | F                      | 0    |          |    |        |  |
| G17                   | G90 | S                 | 1500  | G17               | G90     | S                      | 0    |          |    |        |  |
| G90                   | G98 | M                 | 05,09 | G90               | G98     | M                      | 30   |          |    |        |  |
| G94                   | G15 | T                 | 0000  | G94               | G15     | T                      | 0000 |          |    |        |  |
| G54                   | G50 | H                 | 0000  | G54               | G50     | H                      | 0000 |          |    |        |  |
| G21                   | G69 | D                 | 0000  | G21               | G69     | D                      | 0000 |          |    |        |  |
| G40                   | G64 | .N                | 262   | G40               | G64     | .N                     | 2    |          |    |        |  |
| DATA ^                |     |                   |       |                   |         |                        |      | 08:51:54 |    |        |  |
|                       |     |                   |       |                   |         |                        |      | PATH: 1  |    | AUTO   |  |
|                       |     |                   |       |                   |         |                        |      | RSTR     |    | RETURN |  |

Fig.10-1-9-1

4 In MDI mode, input modes according to the pre-loaded modal values in Fig.10-1-9-1.



5 Press the key, the control returns to the interruption point by G00 and go on execute the program. This execution can be restarted at any place.

**Note 1** The” (1), (2), (3) “headed the coordinates in figure is the moving sequence for the axes moving to the program restarting position. They are set by parameter

**P376.**

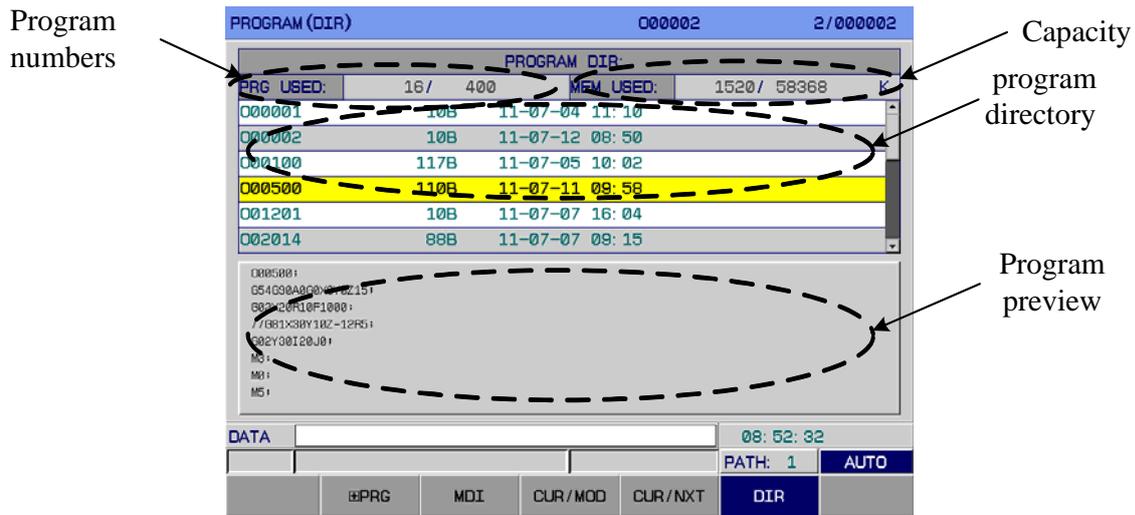
- 2 Check whether the collision occurs when the tool moves to the program restart position, if this possibility exists, move the tool to the place that has no obstructions and restart.
- 3 When the coordinate axis restart the position moving, switch on the single block running, the tool stops each time it finishes an axis movement.
- 4 If there is no absolute position detector, it must restart the line reference point return of advancing after the power is on.
- 5 Don't perform the resetting during the program execution from block research at restarting to restarting, or the restarting must be done from the first step.

## 10.2 Program management

### 10.2.1 Program directory search

Press  key, then press **[DIR]** soft key to enter the program directory page (See

Fig.10-2-1-1) :



**Fig.10-2-1-1**

1) Open the program

Open the program specified: O+sequence number+ENTER key (or EOB key) or sequence number + ENTER key (or EOB key)

In Edit mode, if the sequence number input does not exist, a new program will be created.

2) Deletion of the program:

1. Edit mode Press DEL key to delete the program where cursor locates
2. Edit mode O+ sequence number + DEL or sequence number + DEL

### 10.2.2 Number of the program stored

The maximum number of the programs stored in this system is 400. Look up in Fig. 10-1-2-1

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.10-2-1-1 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 11 CNC program names at most. If the CNC programs are over 10, 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

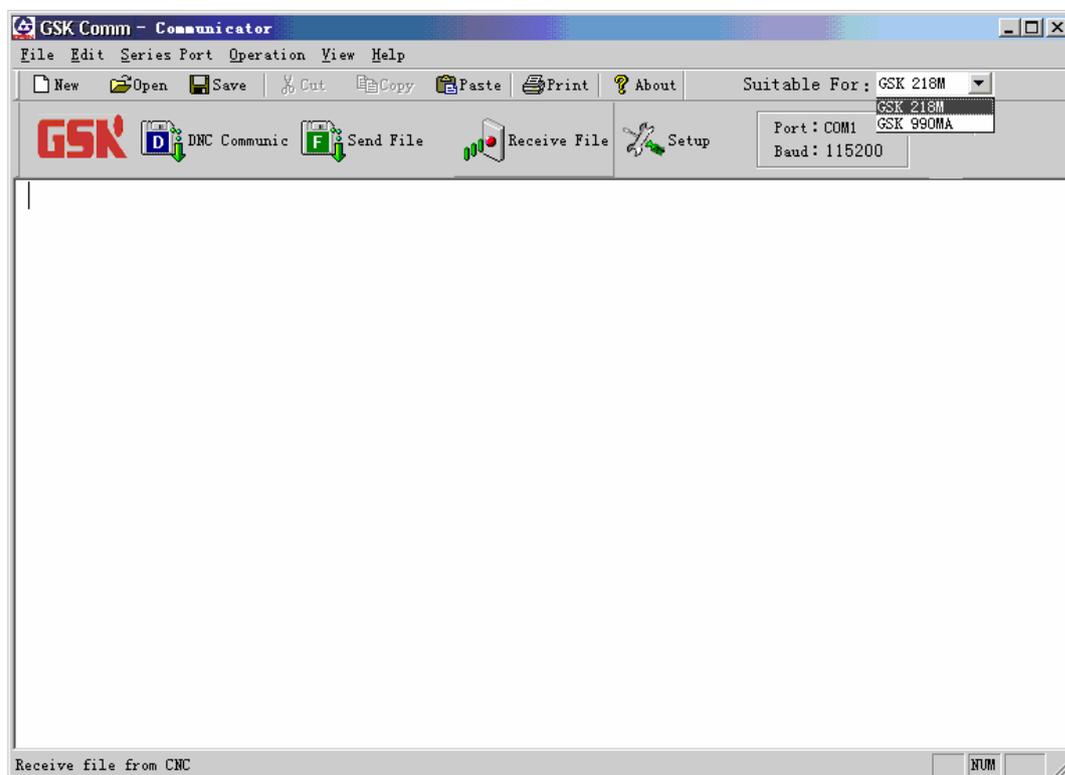
This system can communicate with PC or USB via interface connection.

### 11.1 Serial communication

The serial communication software of this GSK218MC-H system uses Windows interface, which is used to send and receive files, or execute DNC machining from PC terminal to CNC terminal. This software can be run in Win98, WinMe, WinXP or Win2K operation systems.

#### 11.1.1 Program start

Run the CommGSK218MC-H.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, 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, file receiving.
- 5 View menu  
It is used for the tool column display and hiding.
- 6 Help menu  
It is used to view the software version.

### 11.1.3 Software usage

#### 1 DNC transfer

Note: The system I/O channel should be set to 0.

- 1) Open the program file by the“OPEN”button in File menu or the  button in tool column, do a further editing by this software if necessary;
- 2) Open and set the serial port, as shown in above figure, the selected baudrate suitable for GSK218MC-H CNC system default DNC is 38400, which can be reset by the parameter (refer to GSK218MC-H Operation Manual). This 218M system data bits are 8, stop bit is 0 bit, and there is no parity check.
- 3) The sequence of the 1<sup>st</sup> and 2<sup>nd</sup> step can be exchanged which doesn't affect the following transfer and machining; but the following steps must be operated by sequence, or the transfer and machining will be affected.

- 4) As the CNC system and machine are ready, press the  key on panel;
- 5) Open the “DNC”item in Transfer type menu or press the DNC transfer button



in tool column to find out the program needed and transfer data;

- 6) As “Sent Bytes” stops, press the  key on panel to receive data, then press



key to start running;

- 7) Then operate by normal machining pattern;
- 8) During the transmission, the transfer information involving the file names, bytes, lines transferred and the transmission time and speed (bytes/s) will be displayed, which is shown as following:





Fig. 11-1-3-3

The program files can only be sent to “user part”, while the system configuration and backup files can only be sent to system partition, or they won't be identified by system.. To send system configuration and backup files requires the machine builder or dealer level authority, you can enter the relevant password in CNC “password” setting page.

- 5) “Open file” dialogue block will appear after partition selection, press and hold SHIFT or CTRL key to select multiple files, the maximum 399 files can be selected;
- 6) Click “Open” button to return to “Send file” dialogue block after the file is selected;
- 7) The name of the program file sent to user partition should be headed with letter “O”, followed with a number within 5 digits (including 5). Or the following dialogue block will pop up to prompt you to alter the program name:



Fig. 11-1-3-4

- 8) After returning to “Send file” dialogue block, click “Send” button, the file sending will be on, and the following dialogue block will be popped up:

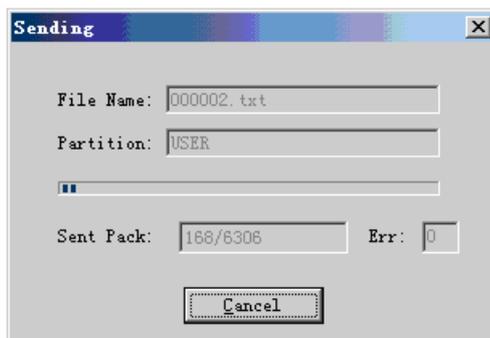


Fig. 11-1-3-5

- 9) Transmission is over.

**Note 1 The system can't send files in DNC mode.**

**2 The operation to transfer program during the system running will affect the system working.**

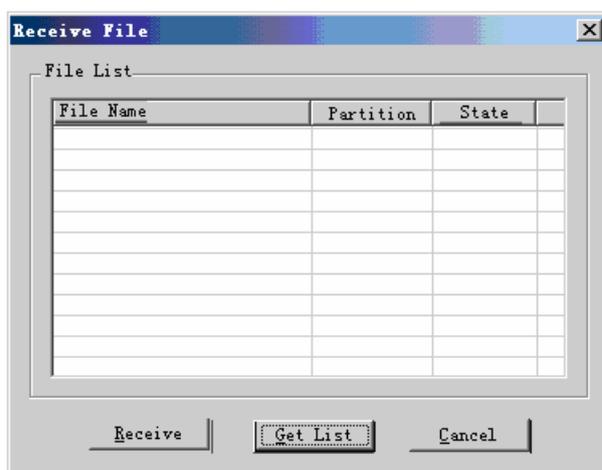
**3 Transfer type for receiving files**

- 1) Enter into the **【DATA】** page in <SETTING> interface, move the red sign to the corresponding box in the "data input" page via direction key according to the transferring content, then press ENTER key.
- 2) Open and set the serial port with a fixed baudrate 115200, the data bits, stop bit and parity check are identical with that in DNC transmission and it can't be changed.

- 3) Open the "Receive file" item of transfer type menu or press the

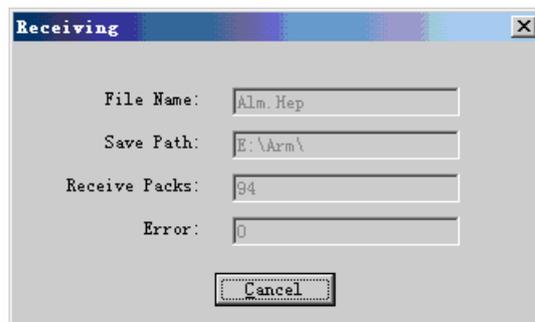


button in tool column, the following dialogue block will pop up:



**Fig. 11-1-3-6**

- 4) Click "Obtain directory" button, the files in CNC system will be listed;
- 5) Select the files to be transferred, multiple files can be selected by pressing and holding SHIFT or CTRL key;
- 6) Click "Start receiving" button for receiving, and the following dialogue block will be popped up;



**Fig. 11-1-3-7**

- 7) Transmission is over.

**Note The system can't receive files in DNC mode.**

## 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 flash disk entry

- 1 Enter into the **【DATA】**page in <SETTING> page, move the cursor to “CNC part program”, in <MDI> mode press ENTER key, then the USB flash disk system begins to start, see details in *OPERATION* Section 3.5.6.
- 2 After entering into USB flash disk program, insert USB flash disk.

### 11.2.3 USB part program operation steps

- 1 To copy CNC programs from USB flash disk to system 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 USB flash disk;
  - c) Press **【COPY】** soft key, it prompts at the page bottom “Are you sure to copy this file to system disk?”, press <CANCEL> key to cancel the copy; 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”.

**Note** The operator should note that if there is a file with the same name in CNC storage disk, this file will be covered.

- 2 To delete files from USB flash 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 USB flash disk;
  - c) Press **【DELETE】** soft key, it prompts at the page bottom “Are you sure to delete 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 USB flash disk file display.
- 3 To copy CNC program from system user disk to USB 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 (such as if the new file name has been input in the new file name box, it will cancel the name input).

**Note** The operator should note that if there is a file with the same name in USB flash disk, it will prompts “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 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.

#### **11.2.4 DNC processing operation steps**

- 1 After CNC system start, set I/O channel value for 1 in <SETTING> page; see details in *OPERATION* Section 3.5.1.
- 2 Insert the USB 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.

#### **11.2.5 USB flash disk system exit**

- 1 Pull out USB flash disk as the indicator for USB flash disk doesn't blink;
- 2 Press **【RETURN】** soft key to return to **【DATA】** soft page in <SETTING> page.

#### **11.2.6 Remarks for USB flash disk model**

Due to the variation of models and drive chips for USB flash disk in market, this CNC system is incapable of identifying all USB flash disks at present. So the USB flash disks shown in the following table are usable via test for this CNC system, but others not shown are untested.

| Company        | Model   | Product standard | Capacity | Chip |
|----------------|---------|------------------|----------|------|
| Tsinghua e era | Mini    | USB2.0           | 128M     |      |
| Lenovo         | T108    | USB2.0           | 128M     |      |
| Lenovo         | B210    | USB2.0           | 256M     |      |
| Lenovo         |         | USB2.0           | 256M     |      |
| Manna          |         | USB2.0           | 128M     |      |
| Aigo           | Special | USB2.0           | 1G       |      |

## APPENDIX 1

### GSK218MC-H 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. |

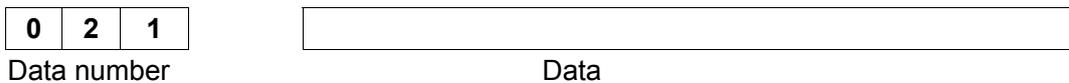
- 1 For bit and axis parameters, the data are comprised by 8 bits with each bit having different meaning.
- 2 The data value range in above table is the common effective range. The specific parameter value range actually differs. See the parameter explanation for details.

**Example**

(1) Meaning of the bit type parameters



(2) Meaning of parameters other than the bit and axis type



- 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 | 3 |  |  | DIR5 | 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.

**Appendix**

- DIRX** =1: X axis feeding direction  
 =0: X axis feeding direction reversing
- DIRY** =1: Y axis feeding direction  
 =0: Y axis feeding direction reversing
- DIRZ** =1: Z axis feeding direction  
 =0: Z axis feeding direction reversing
- DIR4** =1: 4<sup>th</sup> axis feeding direction  
 =0: 4<sup>th</sup> axis feeding direction reversing
- DIR5** =1: 5<sup>th</sup> axis feeding direction  
 =0: 5<sup>th</sup> axis feeding direction reversing

Standard setting: 0 0 1 1 1 1 1 0

System parameter number

|   |   |   |  |  |  |            |            |            |  |            |
|---|---|---|--|--|--|------------|------------|------------|--|------------|
| 0 | 0 | 4 |  |  |  | <b>XIK</b> | <b>AZR</b> | <b>SFD</b> |  | <b>JAX</b> |
|---|---|---|--|--|--|------------|------------|------------|--|------------|

- 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 | <b>IPR</b> |  |  |  |  |  | <b>ISC</b> |  |
|---|---|---|------------|--|--|--|--|--|------------|--|

- ISC** =1: Min. moving unit of 0.0001mm,0.0001deg  
 =0: Min. moving unit of 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 | EDN | EDP |  |  |  | ZRN |
|---|---|---|------|------|-----|-----|--|--|--|-----|

- ZRN** =1: System alarms if instruction other than G28 is specified during auto running.  
 =0: System doesn't alarm if instruction other than G28 is specified during auto running.
- EDP** =1: Rapid traverse and cutting effective of each axis external positive deceleration signal  
 =0: Rapid feed effective of each axis external positive deceleration signal
- EDN** =1: Rapid traverse and cutting effective of each axis external negative deceleration signal  
 =0: Rapid feed effective of each axis external negative deceleration signal
- 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 | ZMI5 | ZMI4 | ZMIz | ZMIy | ZMIx |  | AXS5 | AXS4 |
|---|---|---|------|------|------|------|------|--|------|------|

- AXS4** =1: Set 4<sup>th</sup> axis for linear axis  
 =0: Set 4<sup>th</sup> axis for rotary axis
- AXS5** =1: Set 5<sup>th</sup> axis for linear axis  
 =0: Set 5<sup>th</sup> axis for rotary axis
- 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
- ZMI5** =1: Direction setting of 5th axis reference point return: negative  
 =0: Direction setting of 5th axis reference point return: positive
- Standard setting: 1 0 0 0 0 0 0 0

**Appendix**

System parameter number

|   |   |   |  |  |  |  |  |             |             |             |
|---|---|---|--|--|--|--|--|-------------|-------------|-------------|
| 0 | 0 | 8 |  |  |  |  |  | <b>RRLx</b> | <b>RABx</b> | <b>ROAx</b> |
|---|---|---|--|--|--|--|--|-------------|-------------|-------------|

**ROAx** =1: Rotation axis cycle effective  
 =0: Rotation axis cycle ineffective

**RABx** =1: Rotation direction setting of absolute instruction: instruction value sign  
 =0: Rotation direction setting of absolute instruction: near to the target

**RRLx** =1: Moving amount per revolution rounding for relative coordinates  
 =0: Moving amount per revolution not rounding for relative coordinates

Standard setting: 0 0 0 0 0 0 0 0

System parameter number

|   |   |   |  |  |            |  |  |  |            |  |  |
|---|---|---|--|--|------------|--|--|--|------------|--|--|
| 0 | 0 | 9 |  |  | <b>AWK</b> |  |  |  | <b>ZCL</b> |  |  |
|---|---|---|--|--|------------|--|--|--|------------|--|--|

**ZCL** =1: To cancel local coordinate system when performing manual reference point return  
 =0: Not cancel local coordinate system when performing manual reference point return

**AWK** =1: To change display immediately when workpiece origin offset is changed  
 =0: To change next block display when workpiece origin offset is changed

Standard setting: 0 0 0 0 0 0 0 0

System parameter number

|   |   |   |  |  |  |            |            |  |  |  |  |
|---|---|---|--|--|--|------------|------------|--|--|--|--|
| 0 | 1 | 0 |  |  |  | <b>G52</b> | <b>RLC</b> |  |  |  |  |
|---|---|---|--|--|--|------------|------------|--|--|--|--|

**RLC** =1: To cancel local coordinate system after resetting  
 =0: Not cancel local coordinate system after resetting

**G52** =1: To add tool compensation vector at local coordinate system setting  
 =0: Not add tool compensation vector at local coordinate system setting

Standard setting: 0 0 0 0 0 0 0 0

System parameter number

|   |   |   |            |            |  |  |  |  |  |  |             |
|---|---|---|------------|------------|--|--|--|--|--|--|-------------|
| 0 | 1 | 1 | <b>BFA</b> | <b>LZR</b> |  |  |  |  |  |  | <b>OUT2</b> |
|---|---|---|------------|------------|--|--|--|--|--|--|-------------|

**OUT2** =1: Outer area entry of 2<sup>nd</sup> stroke unallowed  
 =0: Inner area entry of 2<sup>nd</sup> stroke 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 nonlinear.

**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 |  |  |  |  |  |  | DLF | 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

**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

**Appendix**

System parameter number

|          |          |          |  |             |             |             |             |            |             |            |
|----------|----------|----------|--|-------------|-------------|-------------|-------------|------------|-------------|------------|
| <b>0</b> | <b>1</b> | <b>5</b> |  | <b>PACD</b> | <b>PIIS</b> | <b>PILS</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.
- PILS** =1: Forecasting interpolation type: circular interpolation  
=0: Forecasting interpolation type: linear interpolation
- PIIS** =1: Overlapping interpolation effective in acceleration/deceleration blocks before forecasting.  
=0: Overlapping interpolation ineffective in acceleration /deceleration blocks before forecasting.
- PACD** =1: Acceleration /deceleration type before forecasting: S  
=0: Acceleration /deceleration type before forecasting: linear

Standard setting: 0 0 0 0 0 0 0 1

System parameter number

|          |          |          |            |  |  |  |  |             |             |             |
|----------|----------|----------|------------|--|--|--|--|-------------|-------------|-------------|
| <b>0</b> | <b>1</b> | <b>6</b> | <b>ALS</b> |  |  |  |  | <b>FLLS</b> | <b>FBLS</b> | <b>FBOL</b> |
|----------|----------|----------|------------|--|--|--|--|-------------|-------------|-------------|

- 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

|          |          |          |             |             |             |  |             |             |             |             |
|----------|----------|----------|-------------|-------------|-------------|--|-------------|-------------|-------------|-------------|
| <b>0</b> | <b>1</b> | <b>7</b> | <b>CPCT</b> | <b>CALT</b> | <b>WLOE</b> |  | <b>HLOE</b> | <b>CLLE</b> | <b>CBLS</b> | <b>CBOL</b> |
|----------|----------|----------|-------------|-------------|-------------|--|-------------|-------------|-------------|-------------|

- 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: MANUAL running type: exponential  
=0: MANUAL 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 | <b>RVCS</b> |  | <b>RBK</b> | <b>FFR</b> |  | <b>RVIT</b> |
|---|---|---|-------------|--|------------|------------|--|-------------|

- 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 | <b>IOV</b> | <b>ALMS</b> | <b>ALMS5</b> | <b>ALMS4</b> | <b>ALMSZ</b> | <b>ALMSY</b> | <b>ALMSX</b> |
|---|---|---|------------|-------------|--------------|--------------|--------------|--------------|--------------|

- ALMX** =1: High level effective of driver alarm  
=0: Low level effective of driver alarm
- ALMY** =1: High level effective of driver alarm  
=0: Low level effective of driver alarm
- ALMZ** =1: High level effective of driver alarm  
=0: Low level effective of driver alarm
- ALM4** =1: High level effective of driver alarm  
=0: Low level effective of driver alarm



- DRC** =1: Add tool radius compensation in relative position display.  
 =0: Not add tool radius compensation in relative position display.
- DAL** =1: Add tool length compensation in absolute position display.  
 =0: Not add tool length compensation in absolute position display.
- DAC** =1: Add tool radius compensation in absolute position display.  
 =0: Not add tool radius compensation in absolute position display.

Standard setting: 0 0 0 0 0 0 0 0

System parameter number

|   |   |   |  |             |  |            |  |            |  |  |
|---|---|---|--|-------------|--|------------|--|------------|--|--|
| 0 | 2 | 3 |  | <b>POSM</b> |  | <b>SUK</b> |  | <b>DNC</b> |  |  |
|---|---|---|--|-------------|--|------------|--|------------|--|--|

- DNC** =1: To clear DNC running program display by pressing reset key  
 =0: Not clear DNC running program display by pressing reset key
- SUK** =1: To display program list by program numbers.  
 =0: To display program list by logging time.
- 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 | <b>RHD</b> | <b>NPA</b> |  |  | <b>SGD</b> |  | <b>SPS</b> | <b>SVS</b> |
|---|---|---|------------|------------|--|--|------------|--|------------|------------|

- SVS** =1: To display servo setting page.  
 =0: Not display servo setting page.
- SPS** =1: To display spindle setting page.  
 =0: Not display spindle setting page.
- SGD** =1: To display servo wave.  
 =0: Not display servo wave.
- NPA** =1: To switch to alarm page when alarm occurs.  
 =0: Not switch to alarm page when alarm occurs.
- RHD** =1: To update the relative position display at MPG interruption.  
 =0: Not update the relative position display at MPG interruption.

Standard setting: 0 0 0 0 0 0 0 0

System parameter number

|   |   |   |            |            |            |            |            |            |            |            |
|---|---|---|------------|------------|------------|------------|------------|------------|------------|------------|
| 0 | 2 | 5 | <b>ALM</b> | <b>DGN</b> | <b>GRA</b> | <b>SET</b> | <b>OFT</b> | <b>PAR</b> | <b>PRG</b> | <b>POS</b> |
|---|---|---|------------|------------|------------|------------|------------|------------|------------|------------|

- POS** =1: To switchover page by repressing POSITION key in position page.  
 =0: Not switchover page by repressing POSITION key in position page.

**Appendix**

- PRG** =1: To switchover page by repressing PROGRAM key in program page.  
=0: Not switchover page by repressing PROGRAM key in program page.
- PAR** =1: To switchover page by repressing PARAMETER key in parameter page.  
=0: Not switchover page by repressing PARAMETER key in parameter page.
- OFT** =1: To switchover page by repressing OFFSET key in offset page.  
=0: Not switchover page by repressing OFFSET key in offset page.
- SET** =1: To switchover page by repressing SET key in set page.  
=0: Not switchover page by repressing SET key in set page.
- GRA** =1: To switchover page by repressing GRAPHIC key in graphic page.  
=0: Not switchover page by repressing GRAPHIC key in graphic page.
- DGN** =1: To switchover page by repressing DIAGNOSE key in diagnosis page.  
=0: Not switchover page by repressing DIAGNOSE key in diagnosis page.
- ALM** =1: To switchover page by repressing ALARM key in alarm page.  
=0: Not switchover page by repressing ALARM key in alarm page.

Standard setting:     1 1 1 1   1 1 1 1

System parameter number

|          |          |          |             |            |  |  |  |  |             |
|----------|----------|----------|-------------|------------|--|--|--|--|-------------|
| <b>0</b> | <b>2</b> | <b>6</b> | <b>INDX</b> | <b>PMC</b> |  |  |  |  | <b>PETP</b> |
|----------|----------|----------|-------------|------------|--|--|--|--|-------------|

- PETP** =1: To switch to program page by pressing panel Edit key  
=0: Not to switch to program page by pressing panel Edit key
- PMC** =1: To switchover page by repressing PMC key in PMC page.  
=0: Not switchover page by repressing PMC key in PMC page.
- INDX** =1: To switchover page by repressing INDEX key in index page.  
=0: Not switchover page by repressing INDEX key in index page.

Standard setting:     1 1 0 0   0 0 0 1

System parameter number

|          |          |          |  |            |            |            |            |  |            |
|----------|----------|----------|--|------------|------------|------------|------------|--|------------|
| <b>0</b> | <b>2</b> | <b>7</b> |  | <b>PSK</b> | <b>CPD</b> | <b>NE9</b> | <b>OSR</b> |  | <b>NE8</b> |
|----------|----------|----------|--|------------|------------|------------|------------|--|------------|

- NE8** =1: Editing of subprogram with the number 80000 – 89999 unallowed  
=0: Editing of subprogram with the number 80000 – 89999 allowed
- OSR** =1: (O - search) available for program search.  
=0: (O - search) not available for program search.
- NE9** =1: Editing of Subprogram with the number 90000 – 99999 unallowed  
=0: Editing of Subprogram with the number 90000 – 99999 allowed
- CPD** =1: ENTER key needed when deleting programs.  
=0: ENTER key unneeded when deleting programs
- PSK** =1: Search for programs protected effective.  
=0: Search for programs protected ineffective.

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

**Appendix**

System parameter number

|          |          |          |  |            |            |            |            |            |            |            |
|----------|----------|----------|--|------------|------------|------------|------------|------------|------------|------------|
| <b>0</b> | <b>3</b> | <b>1</b> |  | <b>CLR</b> | <b>G13</b> | <b>G91</b> | <b>G19</b> | <b>G18</b> | <b>G17</b> | <b>G01</b> |
|----------|----------|----------|--|------------|------------|------------|------------|------------|------------|------------|

- 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

|          |          |          |  |            |            |  |  |  |  |  |
|----------|----------|----------|--|------------|------------|--|--|--|--|--|
| <b>0</b> | <b>3</b> | <b>2</b> |  | <b>AD2</b> | <b>CIR</b> |  |  |  |  |  |
|----------|----------|----------|--|------------|------------|--|--|--|--|--|

- CIR** =1: Make alarm if distance from start point to center and radius not specified in circular interpolation.  
=0: Do not make alarm if distance from start point to center and radius not specified in circular interpolation.
- 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>EOR</b> |  | <b>M30</b> |  | <b>M02</b> | <b>POL</b> | <b>NOP</b> |
|----------|----------|----------|------------|------------|--|------------|--|------------|------------|------------|

- NOP** =1: Block with only program number, EOB, sequence number ignored  
=0: Block with only program number, EOB, sequence number pre-read
- POL** =1: To program using decimal point.  
=0: To program not using decimal point.
- M02** =1: To return to block beginning when M02 is being executed.  
=0: Not to return to block beginning when M02 is being executed.

- M30** =1: To return to block beginning when M30 is to be executed.  
=0: Not to return to block beginning when M30 is to be executed.
- EOR** =1: To make alarm if “%”occurs in execution.  
=0: To reset if “%”occurs in execution.
- 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

|   |   |   |     |  |  |  |  |  |  |     |
|---|---|---|-----|--|--|--|--|--|--|-----|
| 0 | 3 | 4 | CFH |  |  |  |  |  |  | DWL |
|---|---|---|-----|--|--|--|--|--|--|-----|

- DWL** =1: G04 for dwell per revolution in per revolution feed mode.  
=0: G04 not for dwell per revolution in per revolution feed mode.
- CFH** =1: To clear F,H,D codes at reset or emergency stop.  
=0: To reserve F,H,D codes at reset or emergency stop.

Standard setting: 0 0 0 0 0 0 0 0

System parameter number

|   |   |   |     |     |     |     |     |     |     |  |
|---|---|---|-----|-----|-----|-----|-----|-----|-----|--|
| 0 | 3 | 5 | C07 | C06 | C05 | C04 | C03 | C02 | C01 |  |
|---|---|---|-----|-----|-----|-----|-----|-----|-----|--|

- C01** =1: To clear G codes of 01 group at reset or emergency stop.  
=0: To reserve G codes of 01 group at reset or emergency stop.
- C02** =1: To clear G codes of 02 group at reset or emergency stop.  
=0: To reserve G codes of 02 group at reset or emergency stop.
- C03** =1: To clear G codes of 03 group at reset or emergency stop.  
=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

**Appendix**

System parameter number

|          |          |          |            |            |            |            |            |            |            |            |
|----------|----------|----------|------------|------------|------------|------------|------------|------------|------------|------------|
| <b>0</b> | <b>3</b> | <b>6</b> | <b>C15</b> | <b>C14</b> | <b>C13</b> | <b>C12</b> | <b>C11</b> | <b>C10</b> | <b>C09</b> | <b>C08</b> |
|----------|----------|----------|------------|------------|------------|------------|------------|------------|------------|------------|

- 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

|          |          |          |  |  |  |  |  |  |             |             |
|----------|----------|----------|--|--|--|--|--|--|-------------|-------------|
| <b>0</b> | <b>3</b> | <b>7</b> |  |  |  |  |  |  | <b>WDIR</b> | <b>SCRW</b> |
|----------|----------|----------|--|--|--|--|--|--|-------------|-------------|

- SCRW** =1: To perform pitch compensation.  
=0: Not perform pitch compensation.
- WDIR** =1: Pitch compensation selection: unidirectional  
=0: Pitch compensation selection: bidirectional

Standard setting: 0 0 0 0 0 0 0 0

System parameter number

|          |          |          |            |            |  |  |  |  |  |            |
|----------|----------|----------|------------|------------|--|--|--|--|--|------------|
| <b>0</b> | <b>3</b> | <b>8</b> | <b>PG2</b> | <b>PG1</b> |  |  |  |  |  | <b>SAR</b> |
|----------|----------|----------|------------|------------|--|--|--|--|--|------------|

- 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 |  | EVO |  | EVR |  |  |  | TLC |
|---|---|---|--|-----|--|-----|--|--|--|-----|

**TLC** =1: Tool length compensation type: B

=0: Tool length compensation type: A

**EVR** =1: Offset changed effective by respecifying D in tool radius offset

=0: Offset changed effective in next block in tool radius offset.

**EVO** =1: Offset changed effective by respecifying H in tool length compensation

=0: Offset changed effective in next block in tool length compensation.

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

**Appendix**

System parameter number

|          |          |          |            |            |            |            |  |  |            |            |
|----------|----------|----------|------------|------------|------------|------------|--|--|------------|------------|
| <b>0</b> | <b>4</b> | <b>2</b> | <b>M5B</b> | <b>M5T</b> | <b>RD2</b> | <b>RD1</b> |  |  | <b>EXC</b> | <b>FXY</b> |
|----------|----------|----------|------------|------------|------------|------------|--|--|------------|------------|

**FXY** =1: Axis for drilling canned cycle is the axis selected by program.

=0: Axis for drilling canned cycle is Z.

**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

|          |          |          |  |  |  |  |  |            |            |
|----------|----------|----------|--|--|--|--|--|------------|------------|
| <b>0</b> | <b>4</b> | <b>3</b> |  |  |  |  |  | <b>OZA</b> | <b>SIJ</b> |
|----------|----------|----------|--|--|--|--|--|------------|------------|

**SIJ** =1: Displacement in canned cycle specified by I, J, K.

=0: Displacement in canned cycle specified by Q.

**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 0 0

System parameter number

|          |          |          |  |            |            |            |  |  |            |            |
|----------|----------|----------|--|------------|------------|------------|--|--|------------|------------|
| <b>0</b> | <b>4</b> | <b>4</b> |  | <b>FHD</b> | <b>PCP</b> | <b>DOV</b> |  |  | <b>VGR</b> | <b>G84</b> |
|----------|----------|----------|--|------------|------------|------------|--|--|------------|------------|

**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

**Appendix**

System parameter number

|          |          |          |  |  |  |  |  |  |  |            |            |
|----------|----------|----------|--|--|--|--|--|--|--|------------|------------|
| <b>0</b> | <b>4</b> | <b>8</b> |  |  |  |  |  |  |  | <b>PD1</b> | <b>MDL</b> |
|----------|----------|----------|--|--|--|--|--|--|--|------------|------------|

**MDL** =1: G codes of unidirectional positioning set for modal

=0: G codes of unidirectional positioning not set for modal.

**PD1** =1: To perform in-position check for unidirectional positioning.

=0: Not perform in-position check for unidirectional positioning.

Standard setting: 0 0 0 0 0 0 0 0

System parameter number

|          |          |          |            |            |  |            |            |            |            |            |
|----------|----------|----------|------------|------------|--|------------|------------|------------|------------|------------|
| <b>0</b> | <b>5</b> | <b>0</b> | <b>IDX</b> | <b>SIM</b> |  | <b>G90</b> | <b>INC</b> | <b>ABS</b> | <b>REL</b> | <b>DOP</b> |
|----------|----------|----------|------------|------------|--|------------|------------|------------|------------|------------|

**DOP** =1: Use calculator for indexing table decimal point input

=0: Not use calculator for indexing table decimal point input

**REL** =1: Relative position display setting of indexing table: within 360°

=0: Relative position display setting of indexing table: beyond 360°

**ABS** =1: Use 360°rotation for indexing table absolute coordinate.

=0: Not use 360°rotation for indexing table absolute coordinate.

**INC** =1: Select the latest rotation direction.

=0: Not select the latest rotation direction.

**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.

**IDX** =1: B type by indexing sequence of indexing table.

=0: A type by indexing sequence of indexing table.

Standard setting: 0 1 0 0 0 0 0 0

System parameter number

|          |          |          |  |  |            |  |  |  |  |            |
|----------|----------|----------|--|--|------------|--|--|--|--|------------|
| <b>0</b> | <b>5</b> | <b>1</b> |  |  | <b>SBM</b> |  |  |  |  | <b>G67</b> |
|----------|----------|----------|--|--|------------|--|--|--|--|------------|

**G67** =1: To make alarm if macro instructions cancelled by non-macro modal instructions.

=0: Do not make alarm if macro instructions cancelled by non-macro modal instructions.

**SBM** =1: Single block allowed in macro statement.

=0: Single block unallowed 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 |  |  |  |  | LAD3 | LDA2 | LAD1 | LAD0 |
|---|---|---|--|--|--|--|------|------|------|------|

- LAD0~LAD3** They are binary combination parameters. If they are 0, it uses No.0 ladder; if they are 1~15, it uses No. 1~15 ladders.

Standard setting: 0 0 0 0 0 0 0 1

System parameter number

|   |   |   |  |     |  |  |  |  |  |  |
|---|---|---|--|-----|--|--|--|--|--|--|
| 0 | 5 | 4 |  | ZNM |  |  |  |  |  |  |
|---|---|---|--|-----|--|--|--|--|--|--|

- ZNM** =1: To amplify the center and override display.  
 =0: Not to amplify the center and override display.

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 |  |  |  | HCL | IHD |  | HPF |
|---|---|---|------|--|--|--|-----|-----|--|-----|

- HPF** =1: To select full running for MPG moving.  
 =0: Not select full running for MPG moving.
- IHD** =1: MPG moving is output unit.  
 =0: MPG moving is input unit.
- HCL** =1: Clearing MPG interruption display by soft keys enabled.  
 =0: Clearing MPG interruption display by soft keys disabled.
- 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

**Appendix**

System parameter number

|          |          |          |             |            |            |  |  |  |  |            |
|----------|----------|----------|-------------|------------|------------|--|--|--|--|------------|
| <b>0</b> | <b>5</b> | <b>7</b> | <b>MMDI</b> | <b>OP7</b> | <b>OP6</b> |  |  |  |  | <b>OP1</b> |
|----------|----------|----------|-------------|------------|------------|--|--|--|--|------------|

**OP1** =1: Mode selection by soft keys enabled.

=0: Mode selection by soft keys disabled.

**OP6** =1: Block skip, single block, machine lock, and dry run operation by soft keys enabled.

=0: Block skip, single block, machine lock, and dry run operation by soft keys disabled.

**OP7** =1: Cycle start and dwell operation by soft keys enabled.

=0: Cycle start and dwell operation by soft keys disabled.

**MMDI** =1: Panel keyboard can be replaced by soft keyboard.

=0: Panel keyboard can not be replaced by soft keyboard.

Standard setting: 0 0 0 0 0 0 0 0

System parameter number

|          |          |          |            |            |  |  |  |  |  |
|----------|----------|----------|------------|------------|--|--|--|--|--|
| <b>0</b> | <b>5</b> | <b>8</b> | <b>MOU</b> | <b>MOA</b> |  |  |  |  |  |
|----------|----------|----------|------------|------------|--|--|--|--|--|

**MOA** =1: Outputting all when program restarts.

=0: Outputting the last M, S, T, B codes when program restarts.

**MOU** =1: To output M,S,T,B codes when program restarts.

=0: Not output M, S, T, B codes when program restarts.

Standard setting: 0 0 0 0 0 0 0 0

System parameter number

|          |          |          |             |             |             |  |            |            |             |             |
|----------|----------|----------|-------------|-------------|-------------|--|------------|------------|-------------|-------------|
| <b>0</b> | <b>5</b> | <b>9</b> | <b>OTOP</b> | <b>LEDT</b> | <b>LOPT</b> |  | <b>AOV</b> | <b>DEC</b> | <b>RHPG</b> | <b>OHPG</b> |
|----------|----------|----------|-------------|-------------|-------------|--|------------|------------|-------------|-------------|

**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 |  |  | <b>SCL</b> | <b>SPK</b> | <b>IXC</b> |  |  | <b>TLF</b> |
|---|---|---|--|--|------------|------------|------------|--|--|------------|

**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>SYC</b> |  |  | <b>SSC</b> |
|---|---|---|-------------|-------------|-------------|-------------|------------|--|--|------------|

**SSC** =1: To 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.

**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 USB flash disk.

|      |  |       |
|------|--|-------|
| 0001 | Baudrate of communication channel (DNC ) | 38400 |
|------|--|-------|

Setting range:            0~115200 (unit: BPS)

|      |   |        |
|------|---|--------|
| 0002 | Baudrate of communication channel (file transfer) | 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:            3~4

|      |                                  |   |
|------|----------------------------------|---|
| 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)

|      |  |        |
|------|--|--------|
| 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)

|      |  |        |
|------|--|--------|
| 0014 | External workpiece origin offset amount along 5th 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)

|      |   |        |
|------|---|--------|
| 0019 | Origin offset amount of workpiece coordinate system 1 (G54_5TH) | 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)

**Appendix**

|      |  |        |
|------|--|--------|
| 0024 | Origin offset amount of workpiece coordinate system<br>2 (G55_5TH) | 0.0000 |
|------|--|--------|

Setting range: -9999.9999 ~ 9999.9999 (mm)

|      |  |        |
|------|--|--------|
| 0025 | Origin offset amount of workpiece coordinate system<br>3 (G56_X) | 0.0000 |
|------|--|--------|

Setting range: -9999.9999 ~ 9999.9999 (mm)

|      |  |        |
|------|--|--------|
| 0026 | Origin offset amount of workpiece coordinate system<br>3 (G56_Y) | 0.0000 |
|------|--|--------|

Setting range: -9999.9999 ~ 9999.9999 (mm)

|      |  |        |
|------|--|--------|
| 0027 | Origin offset amount of workpiece coordinate system<br>3 (G56_Z) | 0.0000 |
|------|--|--------|

Setting range: -9999.9999 ~ 9999.9999 (mm)

|      |  |        |
|------|--|--------|
| 0028 | Origin offset amount of workpiece coordinate system<br>3 (G56_4TH) | 0.0000 |
|------|--|--------|

Setting range: -9999.9999 ~ 9999.9999 (mm)

|      |  |        |
|------|--|--------|
| 0029 | Origin offset amount of workpiece coordinate system<br>3 (G56_5TH) | 0.0000 |
|------|--|--------|

Setting range: -9999.9999 ~ 9999.9999 (mm)

|      |  |        |
|------|--|--------|
| 0030 | Origin offset amount of workpiece coordinate system<br>4 (G57_X) | 0.0000 |
|------|--|--------|

Setting range: -9999.9999 ~ 9999.9999 (mm)

|      |  |        |
|------|--|--------|
| 0031 | Origin offset amount of workpiece coordinate system<br>4 (G57_Y) | 0.0000 |
|------|--|--------|

Setting range: -9999.9999 ~ 9999.9999 (mm)

|      |  |        |
|------|--|--------|
| 0032 | Origin offset amount of workpiece coordinate system<br>4 (G57_Z) | 0.0000 |
|------|--|--------|

Setting range: -9999.9999 ~ 9999.9999 (mm)

|      |  |        |
|------|--|--------|
| 0033 | Origin offset amount of workpiece coordinate system<br>4 (G57_4TH) | 0.0000 |
|------|--|--------|

Setting range: -9999.9999 ~ 9999.9999 (mm)

|      |  |        |
|------|--|--------|
| 0034 | Origin offset amount of workpiece coordinate system<br>4 (G57_5TH) | 0.0000 |
|------|--|--------|

Setting range: -9999.9999 ~ 9999.9999 (mm)

|      |  |        |
|------|--|--------|
| 0035 | Origin offset amount of workpiece coordinate system<br>5 (G58_X) | 0.0000 |
|------|--|--------|

Setting range: -9999.9999~9999.9999 (mm)

|      |  |        |
|------|--|--------|
| 0036 | Origin offset amount of workpiece coordinate system<br>5 (G58_Y) | 0.0000 |
|------|--|--------|

Setting range: -9999.9999~9999.9999 (mm)

|      |  |        |
|------|--|--------|
| 0037 | Origin offset amount of workpiece coordinate system<br>5 (G58_Z) | 0.0000 |
|------|--|--------|

Setting range: -9999.9999~9999.9999 (mm)

|      |  |        |
|------|--|--------|
| 0038 | Origin offset amount of workpiece coordinate system<br>5 (G58_4TH) | 0.0000 |
|------|--|--------|

Setting range: -9999.9999~9999.9999 (mm)

|      |  |        |
|------|--|--------|
| 0039 | Origin offset amount of workpiece coordinate system<br>5 (G58_5TH) | 0.0000 |
|------|--|--------|

Setting range: -9999.9999~9999.9999 (mm)

|      |  |        |
|------|--|--------|
| 0040 | Origin offset amount of workpiece coordinate system<br>6 (G59_X) | 0.0000 |
|------|--|--------|

Setting range: -9999.9999~9999.9999 (mm)

|      |  |        |
|------|--|--------|
| 0041 | Origin offset amount of workpiece coordinate system<br>6 (G59_Y) | 0.0000 |
|------|--|--------|

Setting range: -9999.9999~9999.9999 (mm)

|      |  |        |
|------|--|--------|
| 0042 | Origin offset amount of workpiece coordinate system<br>6 (G59_Z) | 0.0000 |
|------|--|--------|

Setting range: -9999.9999~9999.9999 (mm)

|      |  |        |
|------|--|--------|
| 0043 | Origin offset amount of workpiece coordinate system<br>6 (G59_4TH) | 0.0000 |
|------|--|--------|

Setting range: -9999.9999~9999.9999 (mm)

|      |  |        |
|------|--|--------|
| 0044 | Origin offset amount of workpiece coordinate system<br>6 (G59_5TH) | 0.0000 |
|------|--|--------|

Setting range: -9999.9999~9999.9999 (mm)

|      |   |        |
|------|---|--------|
| 0045 | X coordinate of the 1st reference point in machine<br>coordinate system | 0.0000 |
|------|---|--------|

Setting range: -9999.9999~9999.9999 (mm)

**Appendix**

|      |  |        |
|------|--|--------|
| 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)

|      |  |        |
|------|--|--------|
| 0049 | 5TH 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)

|      |  |        |
|------|--|--------|
| 0054 | 5TH 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)

|      |  |        |
|------|--|--------|
| 0059 | 5TH 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)

|      |  |        |
|------|--|--------|
| 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)

|      |  |        |
|------|--|--------|
| 0064 | 5TH 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)

**Appendix**

|      |   |       |
|------|---|-------|
| 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)

|      |   |       |
|------|---|-------|
| 0074 | Negative 5TH axis stroke coordinate of storage travel detection 1 | -9999 |
|------|---|-------|

Setting range: -9999.9999 ~ 9999.9999 (mm)

|      |   |      |
|------|---|------|
| 0075 | Positive 5TH 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)

|      |   |       |
|------|---|-------|
| 0084 | Negative 5TH axis stroke coordinate of storage travel detection 2 | -9999 |
|------|---|-------|

Setting range: -9999.9999~0.0000 (mm)

|      |   |      |
|------|---|------|
| 0085 | Positive 5TH 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)

|      |                              |     |
|------|------------------------------|-----|
| 0087 | Cutting feedrate at power-on | 300 |
|------|------------------------------|-----|

Setting range: 0~9999 (mm/min)

|      |                                |      |
|------|--------------------------------|------|
| 0088 | Rapid traverse speed of X axis | 5000 |
|------|--------------------------------|------|

Setting range: 0~9999 (mm/min)

|      |                                |      |
|------|--------------------------------|------|
| 0089 | Rapid traverse speed of Y axis | 5000 |
|------|--------------------------------|------|

Setting range: 0~9999 (mm/min)

|      |                                |      |
|------|--------------------------------|------|
| 0090 | Rapid traverse speed of Z axis | 5000 |
|------|--------------------------------|------|

Setting range: 0~9999 (mm/min)

**Appendix**

|      |                                  |      |
|------|----------------------------------|------|
| 0091 | Rapid traverse speed of 4TH axis | 5000 |
|------|----------------------------------|------|

Setting range: 0~9999 (mm/min)

|      |                                  |      |
|------|----------------------------------|------|
| 0092 | Rapid traverse speed of 5TH axis | 5000 |
|------|----------------------------------|------|

Setting range: 0~9999 (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: 0~9999(mm/min) Maximum control speed in non-forecast mode

|      |                                 |   |
|------|---------------------------------|---|
| 0095 | Minimum feedrate (for all axes) | 0 |
|------|---------------------------------|---|

Setting range: 0~9999(mm/min) Minimum control speed in non-forecast mode

|      |  |      |
|------|--|------|
| 0096 | Maximum speed in forecasting control mode (for all axes) | 6000 |
|------|--|------|

Setting range: 0~9999(mm/min)

|      |  |   |
|------|--|---|
| 0097 | Minimum speed in forecasting control mode (for all axes) | 0 |
|------|--|---|

Setting range: 0~9999(mm/min)

|      |  |      |
|------|--|------|
| 0098 | Feedrate of manual continuous feed for axes (MANUAL) | 2000 |
|------|--|------|

Setting range: 0~9999 (mm/min)

|      |  |    |
|------|--|----|
| 0099 | Speed(FL) of reference point return (for all axes) | 40 |
|------|--|----|

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)

|                                |  |      |
|--------------------------------|--|------|
| 0104                           | 5 <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)      |  |      |
| 0109                           | L type time constant of pre-acceleration /deceleration of rapid 5TH 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)      |  |      |
| 0113                           | S type time constant of pre-acceleration /deceleration of rapid 4TH axis | 100  |
| Setting range: 0~400 (ms)      |  |      |
| 0114                           | S type time constant of pre-acceleration /deceleration of rapid 5TH axis | 100  |
| Setting range: 0~400 (ms)      |  |      |

**Appendix**

|      |   |    |
|------|---|----|
| 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)

|      |   |    |
|------|---|----|
| 0119 | L type time constant of post acceleration /deceleration of rapid 5TH 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)

|      |   |    |
|------|---|----|
| 0124 | E type time constant of post acceleration /deceleration of rapid 5TH 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)

|      |   |    |
|------|---|----|
| 0137 | Exponential time constant of the post acceleration /deceleration in forecasting control | 60 |
|------|---|----|

Setting range: 0~400 (ms)

**Appendix**

|      |   |    |
|------|---|----|
| 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 circular formation in forecasting control | 5 |
|------|---|---|

Setting range: 0~30

|      |  |    |
|------|--|----|
| 0143 | Angular condition of circular 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 MANUAL feed | 100 |
|------|---|-----|

Setting range: 0~400 (ms)

|      |  |     |
|------|--|-----|
| 0157 | Exponential acceleration /deceleration time constant of axes MANUAL feed | 120 |
|------|--|-----|

Setting range: 0~400 (ms)

|      |   |   |
|------|---|---|
| 0160 | Multiplication coefficient of X axis instruction(CMR) | 1 |
|------|---|---|

Setting range: 1~256

|      |  |   |
|------|--|---|
| 0161 | Multiplication coefficient of Y axis instruction (CMR) | 1 |
|------|--|---|

Setting range: 1~256

|      |  |   |
|------|--|---|
| 0162 | Multiplication coefficient of Z axis instruction (CMR) | 1 |
|------|--|---|

Setting range: 1~256

|      |  |   |
|------|--|---|
| 0163 | Multiplication coefficient of 4TH axis instruction (CMR) | 1 |
|------|--|---|

Setting range: 1~256

**Appendix**

|      |  |   |
|------|--|---|
| 0164 | Multiplication coefficient of 5TH axis instruction (CMR) | 1 |
|------|--|---|

Setting range: 1~256

|      |   |   |
|------|---|---|
| 0165 | Frequency dividing coefficient of X axis instruction(CMD) | 1 |
|------|---|---|

Setting range: 1~256

|      |   |   |
|------|---|---|
| 0166 | Frequency dividing coefficient of Y axis instruction(CMD) | 1 |
|------|---|---|

Setting range: 1~256

|      |   |   |
|------|---|---|
| 0167 | Frequency dividing coefficient of Z axis instruction(CMD) | 1 |
|------|---|---|

Setting range: 1~256

|      |   |   |
|------|---|---|
| 0168 | Frequency dividing coefficient of 4TH axis instruction(CMD) | 1 |
|------|---|---|

Setting range: 1~256

|      |   |   |
|------|---|---|
| 0169 | Frequency dividing coefficient of 5TH axis instruction(CMD) | 1 |
|------|---|---|

Setting range: 1~256

|      |                           |        |
|------|---------------------------|--------|
| 0170 | Servo loop gain of X axis | 0.0000 |
|------|---------------------------|--------|

Setting range: 0~9999.9999

|      |                           |        |
|------|---------------------------|--------|
| 0171 | Servo loop gain of Y axis | 0.0000 |
|------|---------------------------|--------|

Setting range: 0~9999.9999

|      |                           |        |
|------|---------------------------|--------|
| 0172 | Servo loop gain of Z axis | 0.0000 |
|------|---------------------------|--------|

Setting range: 0~9999.9999

|      |                             |        |
|------|-----------------------------|--------|
| 0173 | Servo loop gain of 4TH axis | 0.0000 |
|------|-----------------------------|--------|

Setting range: 0~9999.9999

|      |                             |        |
|------|-----------------------------|--------|
| 0174 | Servo loop gain of 5TH axis | 0.0000 |
|------|-----------------------------|--------|

Setting range: 0~9999.9999

|      |                                   |        |
|------|-----------------------------------|--------|
| 0175 | In-position width of X axis servo | 0.0000 |
|------|-----------------------------------|--------|

Setting range: 0~9999.9999 (mm)

|                                   |  |        |
|-----------------------------------|--|--------|
| 0176                              | In-position width of Y axis servo                  | 0.0000 |
| Setting range: 0~9999.9999 (mm)   |  |        |
| 0177                              | In-position width of Z axis servo                  | 0.0000 |
| Setting range: 0~9999.9999 (mm)   |  |        |
| 0178                              | In-position width of 4TH axis servo                | 0.0000 |
| Setting range: 0~9999.9999 (mm)   |  |        |
| 0179                              | In-position width of 5TH axis servo                | 0.0000 |
| Setting range: 0~9999.9999 (mm)   |  |        |
| 0180                              | Cutting feed in-position width setting of axes     | 0.0000 |
| Setting range: 0~9999.9999 (mm)   |  |        |
| 0181                              | Maximum position error allowable for axes moving   | 0.0000 |
| Setting range: 0~9999.9999 (mm)   |  |        |
| 0182                              | Maximum position error allowable for axes stopping | 0.0000 |
| Setting range: 0~9999.9999 (mm)   |  |        |
| 0183                              | Position error limit when axis servo is off        | 0.0000 |
| Setting range: 0~9999.9999 (mm)   |  |        |
| 0184                              | Servo error allowable for reference point return   | 0.0000 |
| Setting range: 0~9999.9999 (mm)   |  |        |
| 0185                              | Axes grid/reference point offset amount            | 0.0000 |
| Setting range: 0~9999.9999 (mm)   |  |        |
| 0186                              | Alarm time for abnormal load detection             | 500    |
| Setting range: 0~9999             |  |        |
| 0186                              | Alarm time for abnormal load detection             | 500    |
| Setting range: 0~9999             |  |        |
| 0189                              | Reverse precision by backlash compensation         | 0.0100 |
| Setting range: 0.0001~1.0000 (mm) |  |        |

Set  $\alpha = p(189) \times 0.0001$ , in reverse feeding, if the feeding of single servo period is over  $\alpha$ , the backlash compensation begins.

**Appendix**

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)

|      |  |        |
|------|--|--------|
| 0194 | Backlash compensation amount of 5TH 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)

|      |  |        |
|------|--|--------|
| 0199 | Compensation step of 5TH axis clearance by fixed frequency | 0.0030 |
|------|--|--------|

Setting range: 0~99.9999 (mm)

|      |  |    |
|------|--|----|
| 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

**Appendix**

|      |   |   |
|------|---|---|
| 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

|      |   |   |
|------|---|---|
| 0220 | Pitch error compensation number of 5TH 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

|      |   |     |
|------|---|-----|
| 0225 | Pitch error compensation points of 5TH 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)

|      |   |   |
|------|---|---|
| 0230 | Pitch error compensation interval of 5TH axis | 5 |
|------|---|---|

Setting range: 0~99.9999 (mm)

|      |   |       |
|------|---|-------|
| 0231 | Pitch error compensation override of X axis | 0.001 |
|------|---|-------|

Setting range: 0~99.9999

|      |   |       |
|------|---|-------|
| 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

|      |   |       |
|------|---|-------|
| 0235 | Pitch error compensation override of 5TH 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)

|      |  |     |
|------|--|-----|
| 0245 | Time of spindle speed in-position signal detection | 200 |
|------|--|-----|

Setting range: 0~1000(ms)

|      |                                 |      |
|------|---------------------------------|------|
| 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)

**Appendix**

|      |   |    |
|------|---|----|
| 0252 | Spindle motor speed of gear 1 — gear 2 shift in tapping cycle | 50 |
|------|---|----|

Setting range: 0~1000 (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: 0~5000 (r/min)

|      |                         |   |
|------|-------------------------|---|
| 0259 | Spindle servo loop gain | 0 |
|------|-------------------------|---|

Setting range: 0~9999.9999

|      |   |   |
|------|---|---|
| 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: 0~99.9999

|      |  |   |
|------|--|---|
| 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)

|      |   |        |
|------|---|--------|
| 0270 | Retraction amount of high-speed peck drilling cycle G73 | 2.0000 |
|------|---|--------|

Setting range: 0~999.9999 (mm)

|      |   |        |
|------|---|--------|
| 0271 | Reserved space amount of canned cycle G83 | 2.0000 |
|------|---|--------|

Setting range: 0~999.9999 (mm)

|      |  |   |
|------|--|---|
| 0272 | 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)

**Appendix**

|      |  |        |
|------|--|--------|
| 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)

|      |  |      |
|------|--|------|
| 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) | 40 |
|------|---|----|

Setting range: 0~400 (ms)

|      |   |    |
|------|---|----|
| 0299 | Linear acceleration/deceleration time constants of spindle and tapping axis(2nd gear) | 40 |
|------|---|----|

Setting range: 0~400 (ms)

|      |   |    |
|------|---|----|
| 0300 | Linear acceleration/deceleration time constants of spindle and tapping axis(3rd gear) | 40 |
|------|---|----|

Setting range: 0~400 (ms)

|      |   |    |
|------|---|----|
| 0302 | Time constants of spindle and tapping axis in retraction (1st gear) | 20 |
|------|---|----|

Setting range: 0~9999 (ms)

|      |   |    |
|------|---|----|
| 0303 | Time constants of spindle and tapping axis in retraction (2nd gear) | 20 |
|------|---|----|

Setting range: 0~9999 (ms)

|      |   |    |
|------|---|----|
| 0304 | Time constants of spindle and tapping axis in retraction (3rd gear) | 20 |
|------|---|----|

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

**Appendix**

|      |  |   |
|------|--|---|
| 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

|      |   |   |
|------|---|---|
| 0322 | Spindle clearance in rigid tapping (3rd gear) | 0 |
|------|---|---|

Setting range: 0~99.9999

|      |   |   |
|------|---|---|
| 0323 | Spindle instruction multiplication coefficient (CMR) (1st gear) | 1 |
|------|---|---|

Setting range: 1~256

|      |   |   |
|------|---|---|
| 0324 | Spindle instruction multiplication coefficient (CMR) (2nd gear) | 1 |
|------|---|---|

Setting range: 1~256

|      |   |   |
|------|---|---|
| 0325 | Spindle instruction multiplication coefficient (CMR) (3rd gear) | 1 |
|------|---|---|

Setting range: 1~256

|      |   |   |
|------|---|---|
| 0326 | Spindle instruction frequency dividing coefficient (CMD) (1st gear) | 1 |
|------|---|---|

Setting range: 1~256

|      |   |   |
|------|---|---|
| 0327 | Spindle instruction frequency dividing coefficient (CMD) (2nd gear) | 1 |
|------|---|---|

Setting range: 1~256

|      |   |   |
|------|---|---|
| 0328 | Spindle instruction frequency dividing coefficient (CMD) (3rd gear) | 1 |
|------|---|---|

Setting range: 1~256

|      |  |   |
|------|--|---|
| 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

**Appendix**

|      |  |   |
|------|--|---|
| 0339 | Direction and overtravel amount of 5TH 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

|      |                                 |   |
|------|---------------------------------|---|
| 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

|      |                                    |   |
|------|------------------------------------|---|
| 0366 | Handwheel sliding amount allowable | 0 |
|------|------------------------------------|---|

Setting range: 0~10

|      |  |        |
|------|--|--------|
| 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)

|      |  |        |
|------|--|--------|
| 0375 | Positioning error allowable for reverse 5TH axis | 0.0050 |
|------|--|--------|

Setting range: 0~99.9999 (mm)

As the set backlash compensation value (P0190---P0194) of an axis is over the reverse positioning allowable error (P0371---P0375) of this axis, the speed at the end point of a single block lowers to minimum speed before this axis backlash compensation begins, which will make the other axes move a small distance in the backlash compensation period, and that will ensure the resultant path deviating the real path least.

**Appendix**

|      |   |       |
|------|---|-------|
| 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

|      |  |   |
|------|--|---|
| 0384 | Referential counter capacity of 5TH 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                                 |        |

**Appendix**

|      |  |  |
|------|--|--|
| 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 lower than R level   |  |
| 0049 | Z level higher 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                                |  |
| 0065 | 5th axis not on the reference point                                |  |
| 0066 | canned cycle must be cancelled before executing G10                |  |
| 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   |  |
| 0085 | communication error   |  |
| 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  |  |
| 0118 | parenthesis nesting error   |  |
| 0119 | M00~M02, M06, M98, M99 ,and M30 can't be in a same block with other M codes |  |
| 0122 | quadruplicate macro-mode calling  |  |
| 0123 | macro unallowed in DNC  |  |
| 0124 | Illegal program end   |  |
| 0125 | macro format error  |  |

**Appendix**

|      |   |  |
|------|---|--|
| 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  |  |
| 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  |  |
| 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 botton 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                               |  |
| 0212 | illegal plane selection                          |  |
| 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                         |  |

**Appendix**

|      |  |  |
|------|--|--|
| 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                  |  |
| 0400 | servo alarm: n-axis overload                     |  |
| 0401 | servo alarm: n-axis VRDY off                     |  |
| 0404 | servo alarm: n-axis VRDY on                      |  |
| 0405 | servo alarm: (zero return error)                 |  |
| 0407 | servo alarm: superheterodyning                   |  |
| 0409 | torque alarm: superheterodyning                  |  |
| 0410 | servo alarm: n-axis superheterodyning            |  |
| 0411 | servo alarm: n-axis superheterodyning            |  |
| 0413 | servo alarm: n-axis LSI overflow                 |  |
| 0414 | servo alarm: n-axis detection error              |  |
| 0415 | servo alarm: n-axis move too fast                |  |
| 0416 | servo alarm: n-axis detecting broken off         |  |
| 0417 | servo alarm: n-axis parameter error              |  |
| 0420 | synch torque: superheterodyning                  |  |
| 0421 | servo alarm: superheterodyning                   |  |
| 0422 | servo alarm: speed error                         |  |
| 0423 | servo alarm: cumulative travel superheterodyning |  |
| 0448 | n-axis: unmatched feedback alarm                 |  |
| 0449 | n-axis: INV.IPM alarm                            |  |
| 0451 | X axis driver alarm                              |  |
| 0452 | Y axis driver alarm                              |  |
| 0453 | Z axis driver alarm                              |  |
| 0454 | 4th axis driver alarm                            |  |

|      |  |  |
|------|--|--|
| 0455 | 5th 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  |  |
| 0740 | rigid tapping alarm: superheterodyning                               |  |
| 0741 | rigid tapping alarm: superheterodyning                               |  |
| 0742 | rigid tapping alarm: LSI overflow                                    |  |
| 0751 | 1st spindle alarm (AL-XX) detected                                   |  |
| 0754 | spindle abnormal torque alarm  |  |
| 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 linage 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                  |  |

**Appendix**

|      |  |  |
|------|--|--|
| 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/LBL wrongly processed or beyond the capacity      |  |
| 1016 | incomplete NET constitution  |  |
| 1017 | unsupported NET constitution exists  |  |
| 1018 | suspended node exists in NET   |  |
| 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 | programming attempt without ROM and ROM                                      |  |
| 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                                      |  |
| 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                         |  |