



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

The User Manual is applied to the following CNC systems:

| Type        | structure  | LCD size | Remark                               |
|-------------|------------|----------|--------------------------------------|
| GSK980TB3   | horizontal | 7.0      | LCD size is defaulted to be 7.0 inch |
| GSK980TB3-V | vertical   | 7.0      | LCD size is defaulted to be 7.0 inch |

GSK980TB3 is upgrading products of GSK980TB2, which can control three feed shaft, a spindle , and add the function of automatic feeding, groove cycle processing(G71) and so on, which can significantly improve the efficiency of machining parts, precision and surface roughness.

As GSK980TB2 upgrades, GSK980TB3 (GSK980TB3 - V) is the best choice of the economical cnc lathe technology upgrading.

## FOREWORD

Dear user,

We are really grateful for your patronage and purchase of CNC systems made by GSK CNC Equipment Co., Ltd.

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

### Warning



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

Please carefully read this user manual before use!

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

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

## Safety Precaution



### **Warning, caution and note**

This manual includes safety precaution for protecting the user and preventing damage to the machine. Precautions are classified into Warning and Caution according to their bearing on safety. Read the Warning, Caution and Note thoroughly before attempting to use the machine.



### **Warning**

There is a danger of the user being injured or the equipment being damaged if the approved procedure is not observed.



### **Caution**

There is a danger of the equipment being damaged if the approved procedure is not observed.

### **Note**

It is used to indicate supplementary information other than Warning and Caution.

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

This manual is reserved by end user.

■ **Delivery and storage**

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

■ **Open packing box to check**

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

■ **Connection**

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

■ **Troubleshooting**

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

## Safety Responsibility

### **Manufacturer's safety responsibility**

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

### **User's safety responsibility**

- The user should study and train the system safety operation, master the safety operation content.
- The user should be responsible for the danger caused by increasing, changing or modifying the CNC system, the accessories by itself.
- The user should be responsible for the danger because of the mistaken operation, regulation, maintenance, installation and storage.

**This user manual shall be kept by final user.**

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

# Contents

## PART ONE SUMMARY

|                                 |          |
|---------------------------------|----------|
| <b>Chapter One Summary.....</b> | <b>3</b> |
| 1.1 Summary.....                | 3        |
| 1.2 Product Brief.....          | 3        |
| 1.3 Model and Meaning.....      | 4        |
| 1.4 Order.....                  | 4        |

## PART TWO PROGRAMMING

|                                                                          |           |
|--------------------------------------------------------------------------|-----------|
| <b>Chapter One Programming Fundamentals .....</b>                        | <b>7</b>  |
| 1.1 Coordinate axis definition.....                                      | 7         |
| 1.2 Machine coordinate system, Machine Zero .....                        | 8         |
| 1.3 Workpiece Coordinate System and Reference Point(Program Zero).....   | 8         |
| 1.4 Controlled axes .....                                                | 9         |
| 1.5 Input Increment.....                                                 | 9         |
| 1.6 Maximum Stroke .....                                                 | 9         |
| 1.7 Absolute Programming and Incremental Programming.....                | 9         |
| 1.8 Diameter and Radius Programming.....                                 | 10        |
| 1.9 Modal, Simple and Initial State.....                                 | 11        |
| <b>Chapter Two Structure of an Part Program.....</b>                     | <b>13</b> |
| 2.1 General Structure of a Program .....                                 | 13        |
| 2.1.1 Program Name .....                                                 | 13        |
| 2.1.2 Sequence Number and Block.....                                     | 14        |
| 2.1.3 Word .....                                                         | 14        |
| 2.2 Relationship between Command Numerical Value and Decimal Point ..... | 16        |
| 2.3 Subprogram.....                                                      | 16        |
| 2.3.1 Main Program and Subprogram.....                                   | 16        |
| 2.3.2 Subprogram Call (M98) .....                                        | 17        |
| 2.3.3 Return from subprogram (M99) .....                                 | 17        |
| <b>Chapter Three Preparatory Function G Command .....</b>                | <b>19</b> |
| 3.1 Category of Preparatory Function G Command .....                     | 19        |
| 3.2 Simple G Command.....                                                | 21        |
| 3.2.1 Rapid Positioning (G00) .....                                      | 21        |
| 3.2.2 Linear Interpolation (G01) .....                                   | 22        |
| 3.2.3 Circular Interpolation (G02/G03) .....                             | 24        |
| 3.2.4 Chamfering/Cornering Arc .....                                     | 26        |
| 3.2.5 Dwell G04.....                                                     | 27        |
| 3.2.6 Programs Specifying Parameter Function (G10) .....                 | 28        |
| 3.2.7 Machine Zero Return (G28) .....                                    | 28        |
| 3.2.8 Workpiece Coordinate System Setting (G50) .....                    | 29        |
| 3.2.9 Workpiece Coordinate System Offset (G51) .....                     | 31        |
| 3.2.10 Feed per minute (G98).....                                        | 32        |
| 3.2.11 Feed per Rev (G99).....                                           | 32        |
| 3.2.12 Constant Surface Control (G96, G97).....                          | 32        |
| 3.2.13 Skip Function (G31) .....                                         | 34        |
| 3.2.14 Thread Cutting with Invariable Pitch (G32) .....                  | 35        |
| 3.2.15 Thread Cutting with Variable Pitch (G34) .....                    | 38        |
| 3.2.16 Tapping Cycle (G33) .....                                         | 38        |
| 3.3 Single Fixed Cycle Command .....                                     | 39        |
| 3.3.1 Outer (Inner) Cutting Cycle (G90) .....                            | 40        |
| 3.3.2 End Cutting Cycle (G94) .....                                      | 42        |

|                      |                                                                 |           |
|----------------------|-----------------------------------------------------------------|-----------|
| 3.3.3                | Thread Cutting Cycle (G92)                                      | 44        |
| 3.3.4                | Notes in Single Fixed Cycle Commands                            | 46        |
| 3.4                  | Compound Fixed Cycle Commands                                   | 47        |
| 3.4.1                | Outer (Inner) Roughing Cycle (G71)                              | 47        |
| 3.4.2                | End Roughing Cycle (G72)                                        | 50        |
| 3.4.3                | Closed Cutting Cycle (G73)                                      | 53        |
| 3.4.4                | Finishing Cycle (G70)                                           | 56        |
| 3.4.5                | End Deep Hole Machining Cycle (G74)                             | 56        |
| 3.4.6                | Outer Grooving Cycle (G75)                                      | 58        |
| 3.4.7                | Compound Thread Cutting Cycle (G76)                             | 60        |
| 3.4.8                | Notes of Compound Fixed Cycle Codes                             | 62        |
| <b>Chapter Four</b>  | <b>Miscellaneous Function M Codes</b>                           | <b>64</b> |
| 4.1                  | Miscellaneous Function (M function)                             | 64        |
| 4.2                  | Special M Codes (M21.M22.M23.M24 )                              | 65        |
| 4.3                  | M Code Calling Subprograms                                      | 66        |
| 4.4                  | S Codes Calling Subprograms                                     | 66        |
| 4.5                  | T Codes Calling Subprograms                                     | 67        |
| 4.6                  | Parameters of Miscellaneous Function                            | 67        |
| 4.7                  | Alarm Related to Miscellaneous Function                         | 67        |
| <b>Chapter Five</b>  | <b>Spindle Function S Code</b>                                  | <b>69</b> |
| 5.1                  | Spindle Switching Value Control                                 | 69        |
| 5.2                  | Spindle Analog Value Control                                    | 69        |
| <b>Chapter Six</b>   | <b>Tool Function T Code</b>                                     | <b>70</b> |
| 6.1                  | Tool change Process                                             | 70        |
| 6.2                  | Relevant Parameters of Tool Change                              | 71        |
| 6.3                  | Tool Offset                                                     | 71        |
| 6.3.1                | Basic Tool Offset                                               | 72        |
| 6.3.2                | T Code for tool offset                                          | 72        |
| 6.3.3                | Offset                                                          | 73        |
| 6.3.4                | Programming Example                                             | 74        |
| 6.3.5                | Single T code                                                   | 75        |
| <b>Chapter Seven</b> | <b>User Macro Program</b>                                       | <b>76</b> |
| 7.1                  | User Macro Code                                                 | 76        |
| 7.2                  | User Macro Program Body                                         | 76        |
| 7.3                  | Operation Command and Transfer Command (G65)                    | 79        |
| 7.4                  | Notes for User Macro Program Body                               | 83        |
| 7.5                  | User Macro Program Example                                      | 83        |
| <b>Chapter Eight</b> | <b>Tool Compensation C Function</b>                             | <b>84</b> |
| 8.1                  | Basic Concept of Tool Compensation C Function                   | 84        |
| 8.1.1                | Imaginary Tool Nose                                             | 84        |
| 8.1.2                | Imaginary Tool Nose Direction                                   | 86        |
| 8.1.3                | Compensation Value Setting                                      | 89        |
| 8.1.4                | Relative Position between Tool and Workpiece                    | 90        |
| 8.1.5                | Inside and outside                                              | 92        |
| 8.1.6                | Command Format of G41, G42, G40                                 | 93        |
| 8.2                  | Concrete Tool Compensation                                      | 93        |
| 8.2.1                | Concrete Path Decomposition of Tool Nose Radius Compensation    | 93        |
| 8.2.2                | Compensation direction change when tool compensation proceeding | 99        |
| 8.2.3                | Temporary Tool Compensation Cancel                              | 101       |
| 8.2.4                | Non-Movement Command when Tool Compensation                     | 103       |
| 8.2.5                | Tool Compensation Interference Check                            | 104       |
| 8.2.6                | Tool Nose Radius Compensation in G90/G94                        | 106       |
| 8.2.7                | Tool Nose Radius Compensation in G70                            | 107       |
| 8.3                  | Notes of Tool Compensation C                                    | 107       |
| 8.4                  | Tool Compensation C Example                                     | 108       |

PART THREE OPERATION

|                                                                                                   |            |
|---------------------------------------------------------------------------------------------------|------------|
| <b>Chapter One Operation Panel .....</b>                                                          | <b>113</b> |
| 1.1 Panel division .....                                                                          | 113        |
| 1.2 Instructions for Panel Functions .....                                                        | 114        |
| 1.2.1 LCD (liquid crystal) display area .....                                                     | 114        |
| 1.2.2 Edit Keypad Area .....                                                                      | 114        |
| 1.2.3 Introduction for Screen Operation Keys .....                                                | 115        |
| 1.2.4 Machine Control Area .....                                                                  | 116        |
| <b>Chapter Two System Power ON/OFF and Safety Operation .....</b>                                 | <b>119</b> |
| 2.1 System Power-on .....                                                                         | 119        |
| 2.2 System Power-off .....                                                                        | 119        |
| 2.3 Safety Operation .....                                                                        | 120        |
| 2.3.1 Reset .....                                                                                 | 120        |
| 2.3.2 Emergency Stop .....                                                                        | 120        |
| 2.3.3 Feed Hold .....                                                                             | 120        |
| 2.3.4 Power-off .....                                                                             | 120        |
| 2.4 Cycle Start and Feed Hold .....                                                               | 120        |
| 2.5 Overtravel Protection .....                                                                   | 121        |
| 2.5.1 Hardware Overtravel Protection .....                                                        | 121        |
| 2.5.2 Software Overtravel Protection .....                                                        | 121        |
| <b>Chapter Three Interface Display and Data Modification and Setting .....</b>                    | <b>122</b> |
| 3.1 Position Display .....                                                                        | 122        |
| 3.1.1 Four Ways for Interface Display .....                                                       | 122        |
| 3.1.2 Display of Machining Time, Parts Number, Programming Speed, Override and Actual Speed ..... | 123        |
| 3.1.3 Clearing Methods of Parts Number and Cutting Time .....                                     | 124        |
| 3.2 Program Display .....                                                                         | 125        |
| 3.3 Display, Modification and Setting of the Tool Offset .....                                    | 126        |
| 3.3.1 Tool Offset Display .....                                                                   | 126        |
| 3.3.2 Tool Offset Modification and Setting .....                                                  | 127        |
| 3.4 Alarm Display .....                                                                           | 127        |
| 3.5 Setting Display .....                                                                         | 128        |
| 3.5.1 Instruction for 【Setting】 Interface Operation .....                                         | 128        |
| 3.5.2 Operating Instructions for 【Graph】 Interface .....                                          | 130        |
| 3.6 Display, Modification and Setting of the Parameter .....                                      | 132        |
| 3.6.1 Parameter Display .....                                                                     | 132        |
| 3.6.2 Modification and Setting for Parameter and Screw Pitch Compensation Value .....             | 133        |
| 3.7 Diagnosis Display .....                                                                       | 135        |
| 3.7.1 Diagnosis Data Display .....                                                                | 135        |
| <b>Chapter Four Manual Operation .....</b>                                                        | <b>138</b> |
| 4.1 Coordinate Axis Movement .....                                                                | 138        |
| 4.1.1 Manual Feed .....                                                                           | 138        |
| 4.1.2 Manual Rapid Traverse .....                                                                 | 138        |
| 4.1.3 Manual Feed and Manual Rapid Traverse Rate Selection .....                                  | 138        |
| 4.2 Spindle Control .....                                                                         | 139        |
| 4.2.1 Spindle CW (Negative rotation) .....                                                        | 139        |
| 4.2.2 Spindle CCW (Positive rotation) .....                                                       | 140        |
| 4.2.3 Spindle Stop .....                                                                          | 140        |
| 4.3 Other Manual Operations .....                                                                 | 140        |
| 4.3.1 Cooling Control .....                                                                       | 140        |
| 4.3.2 Lubricating Control .....                                                                   | 140        |
| 4.3.3 Manual Tool Change .....                                                                    | 140        |
| 4.4 Tool Setting .....                                                                            | 140        |
| 4.4.1 Fixed-point Tool Setting .....                                                              | 141        |
| 4.4.2 Tool Setting by Test Cutting .....                                                          | 142        |
| 4.4.3 Tool Setting by Machine Zero Return .....                                                   | 143        |
| 4.5 Offset Alteration .....                                                                       | 146        |

|                      |                                                         |            |
|----------------------|---------------------------------------------------------|------------|
| <b>Chapter Five</b>  | <b>MPG/Sing Step Operation .....</b>                    | <b>147</b> |
| 5.1                  | Step Feed .....                                         | 147        |
| 5.2                  | MPG (Manual pulse generator) Feed.....                  | 147        |
| 5.2.1                | Moving Axis and Direction Selection .....               | 148        |
| 5.2.2                | Instructions for MPG Feed .....                         | 149        |
| 5.3                  | The Miscellaneous Control in MPG/Step Operation .....   | 149        |
| <b>Chapter Six</b>   | <b>MDI Operation .....</b>                              | <b>150</b> |
| 6.1                  | MDI Blocks Input.....                                   | 150        |
| 6.2                  | MDI Block Operation and Stop.....                       | 150        |
| 6.3                  | MDI Block Words Alteration and Clearing .....           | 151        |
| 6.4                  | Conversion of Operation Modes.....                      | 151        |
| <b>Chapter Seven</b> | <b>Edit Operation .....</b>                             | <b>152</b> |
| 7.1                  | Program Edit.....                                       | 152        |
| 7.1.1                | Program Creation .....                                  | 152        |
| 7.1.2                | Deletion of a program.....                              | 156        |
| 7.1.3                | Deletion of All Programs.....                           | 156        |
| 7.1.4                | Duplication of the Program .....                        | 156        |
| 7.1.5                | Rename of the Program .....                             | 156        |
| 7.2                  | Program Management.....                                 | 157        |
| 7.2.1                | Search of Program List.....                             | 157        |
| 7.2.2                | Quantity of the Saved Programs .....                    | 157        |
| 7.2.3                | Memory Capacity .....                                   | 157        |
| 7.2.4                | Program Lock .....                                      | 158        |
| 7.3                  | Other Operations Available in Edit Mode.....            | 158        |
| <b>Chapter Eight</b> | <b>AUTO Operation .....</b>                             | <b>159</b> |
| 8.1                  | Selection of the Program To Be Run .....                | 159        |
| 8.2                  | Start of the Auto Run .....                             | 159        |
| 8.3                  | Auto Run Stop .....                                     | 160        |
| 8.4                  | Auto Run From an Arbitrary Block .....                  | 160        |
| 8.5                  | Dry Run .....                                           | 161        |
| 8.6                  | Single Block Execution .....                            | 162        |
| 8.7                  | Machine Lock.....                                       | 163        |
| 8.8                  | M. S. T Lock.....                                       | 164        |
| 8.9                  | Block Skip .....                                        | 164        |
| 8.10                 | Ajustment of the Feedrate, Rapid Rate in Auto Mode..... | 164        |
| 8.11                 | Spindle Speed Adjustment in Auto Mode.....              | 165        |
| 8.12                 | Cooling Control .....                                   | 165        |
| <b>Chapter Nine</b>  | <b>Zero Return Operation .....</b>                      | <b>166</b> |
| 9.1                  | Concept of the Machine Zero Return .....                | 166        |
| 9.2                  | The Operation Procedures of Machine Zero Return.....    | 167        |
| 9.3                  | Concept of Program Zero Point.....                      | 168        |
| 9.4                  | The Operation Procedures of Program Zero Return .....   | 168        |
| <b>Chapter Ten</b>   | <b>System Communication .....</b>                       | <b>169</b> |
| 10.1                 | Serial Port Communication.....                          | 169        |
| 10.1.1               | Program Start.....                                      | 169        |
| 10.1.2               | Function Instruction.....                               | 170        |
| 10.1.3               | Software Usage.....                                     | 170        |
| 10.1.4               | Communication between CNC and CNC .....                 | 173        |
| 10.2                 | USB Communication.....                                  | 174        |
| 10.2.1               | Overview and Cautions .....                             | 174        |
| 10.2.2               | Enter U-disk .....                                      | 174        |
| 10.2.3               | Procedures .....                                        | 175        |
| 10.2.4               | U-disk Removal.....                                     | 176        |

PART FOUR CONNECTION

|                                                                                                                   |            |
|-------------------------------------------------------------------------------------------------------------------|------------|
| <b>Chapter One System Configuration and Installation .....</b>                                                    | <b>179</b> |
| 1.1 System Configuration .....                                                                                    | 179        |
| 1.2 System Installation.....                                                                                      | 179        |
| 1.3 Mounting Dimension of the CNC System .....                                                                    | 180        |
| 1.4 Additional Panel .....                                                                                        | 181        |
| <b>Chapter Two Equipment Connection.....</b>                                                                      | <b>183</b> |
| 2.1 System Interfaces.....                                                                                        | 183        |
| 2.1.1 Interface Layout.....                                                                                       | 183        |
| 2.1.2 Explanations for Interfaces .....                                                                           | 184        |
| 2.1.3 Connection Diagram.....                                                                                     | 184        |
| 2.2 Connection of the System to the Drive Unit .....                                                              | 184        |
| 2.2.1 Connection to the Drive Unit.....                                                                           | 184        |
| 2.2.2 Signal Explanation.....                                                                                     | 186        |
| 2.2.3 Connection Between the System and the Drive Unit.....                                                       | 189        |
| 2.3 Connection to the Spindle Encoder.....                                                                        | 190        |
| 2.4 RS232-C Standard Serial Port (Optional) .....                                                                 | 191        |
| 2.5 Spindle Interface .....                                                                                       | 191        |
| 2.5.1 Connection drawing of 980TB3 and normal relay.....                                                          | 193        |
| 2.5.2 Connection drawing of 980TB3 and spindle driver DAP03(require position and speed control function).....     | 194        |
| 2.5.3 Connection drawing of 980TB3 and spindle driver GS3000Y-N (require position and speed control function .... | 195        |
| 2.6 MPG Interface.....                                                                                            | 196        |
| 2.6.1 Normal MPG connetcton drawing .....                                                                         | 196        |
| 2.6.2 Connetion drawing of handwheel (match MPG GSKSC10) .....                                                    | 197        |
| 2.7 Connection of the Power Interface.....                                                                        | 198        |
| <b>Chapter Three I/O Interface of Machine Tool.....</b>                                                           | <b>199</b> |
| 3.1 List of the Machine Tool I/O Interface.....                                                                   | 199        |
| 3.1.2 Expansion I/O Interface of GSK980TB3/TB3-V .....                                                            | 201        |
| 3.2 Input Signal.....                                                                                             | 202        |
| 3.3 Signal Expalnation .....                                                                                      | 203        |
| 3.4 Output Signal .....                                                                                           | 208        |
| 3.5 Spindle Automatic Gearing Control.....                                                                        | 211        |
| 3.6 Chuck Control (Chuck Detection Function) .....                                                                | 212        |
| 3.7 Tailstock Control (Tailstock hierarchical advance/retreat control) .....                                      | 214        |
| 3.8 External Program Lock .....                                                                                   | 216        |
| 3.9 External Feed Pause and Spindle Pause.....                                                                    | 216        |
| 3.10 I/O Signal Disgnosis List.....                                                                               | 217        |
| 3.11 Function Description.....                                                                                    | 218        |
| <b>Chapter Four Machine Tool Debugging.....</b>                                                                   | <b>219</b> |
| 4.1 Debugging Preparation .....                                                                                   | 219        |
| 4.2 System Power On.....                                                                                          | 219        |
| 4.3 Emergency Stop and Limit .....                                                                                | 219        |
| 4.4 Drive Axis Debugging .....                                                                                    | 220        |
| 4.5 Gear Ratio Adjustment.....                                                                                    | 220        |
| 4.6 Rapid Traverse Rate and Linear ACC&DEC Adjustment.....                                                        | 220        |
| 4.7 Upper Speed of the Cutting Feed.....                                                                          | 221        |
| 4.8 Adjustment for Thread Machining.....                                                                          | 221        |
| 4.9 Machine (mechanical) Zero Adjustment.....                                                                     | 221        |
| 4.10 Offset between Lead Screws.....                                                                              | 221        |
| 4.11 Electronic Toolpost Debugging.....                                                                           | 222        |
| 4.12 Spindle Adjustment.....                                                                                      | 223        |
| 4.13 Step/MPG Adjustment .....                                                                                    | 223        |
| 4.14 Three-state Switch Adjustment .....                                                                          | 223        |
| 4.15 Other Adjustment .....                                                                                       | 223        |

|                     |                                                       |            |
|---------------------|-------------------------------------------------------|------------|
| <b>Chapter Five</b> | <b>Memorizing Screw-pitch Error Compensation.....</b> | <b>225</b> |
|---------------------|-------------------------------------------------------|------------|

## APPENDIX

|                     |                                                                |            |
|---------------------|----------------------------------------------------------------|------------|
| <b>Appendix I</b>   | <b>Paremeter List.....</b>                                     | <b>231</b> |
| 1.1                 | Explanations for parameter.....                                | 231        |
| 1.2                 | Bit parameter .....                                            | 231        |
| 1.3                 | Data Parameter .....                                           | 238        |
| <b>Appendix II</b>  | <b>Alarm List.....</b>                                         | <b>253</b> |
| <b>Appendix III</b> | <b>Diagnosis List.....</b>                                     | <b>261</b> |
| 1.                  | DI/DO Diagnosis Message .....                                  | 261        |
| 1.1                 | Input Signal from Machine Side (I/O board).....                | 261        |
| 1.2                 | Signal Input to Machine Side (I/O board) .....                 | 261        |
| 2.                  | CNC Interface Signal .....                                     | 262        |
| 3.                  | Operation Panel Key.....                                       | 263        |
| 4.                  | CNC Input/Output Signal .....                                  | 263        |
| 5.                  | Signals from CNC .....                                         | 264        |
| 6.                  | CNC Internal State Message.....                                | 265        |
| <b>Appendix IV</b>  | <b>Specification List .....</b>                                | <b>267</b> |
| <b>Appendix V</b>   | <b>Wiring for CNC Matching Taiwan LIO SHING Tool Post.....</b> | <b>269</b> |
| <b>Appendix VI</b>  | <b>Factory Parameters .....</b>                                | <b>270</b> |

# PART ONE SUMMARY



## Chapter One Summary

### 1.1 Summary

**Part One Summary** Introduce the system, composition of chapters, the system model, relevant use explanations, and precautions before reading the user manual.

**Part Two Programming** Describe composition of part programs, fundamentals of programming, each code's function, command format, characteristics and restrictions in NC programming, and so on.

**Part Three Operation** Narrate each window and setting of the system, each operation of machine, program input/output, edit, and the system's communication, and so on.

**Part Four Connection** Mention the system's structure, installation dimension, connection among devices, I/O interface definition, machine debugging, the system's specification, optional memory pitch error compensation function.

**Appendix** List parameter tables (including parameter default value and parameter setting range), alarm tables and diagnosis table.

The user manual is applied to **GSK980TB3/ GSK980 TB3-V** CNC systems.

### 1.2 Product Brief

**GSK980TB3 is upgrading products of GSK980TB2, which can control three feed shaft, a spindle, and add the function of automatic feeding, groove cycle processing(G71) and so on, which can significantly improve the efficiency of machining parts, precision and surface roughness. As GSK980TB2 upgrades, GSK980TB3 (GSK980TB3-V) is the best choice of the economical cnc lathe technology upgrading**

As upgraded products, the systems have the following features:

- 32-bit CUP, CPLD hardware interpolation technology to realize high-speed  $\mu\text{m}$  level control;
- 4-layer circuit board with high integrated level, proper system technology structure and high reliability;
- Chinese/English LCD, friendly interface and easy operation;
- Adjustable acceleration/deceleration to be matched with stepper drive unit or servo drive unit
- Changeable electronic gear ratio;
- Prepositioning USB interfaces and RS232 interfaces to be convenient to the user managing programs.

### 1.3 Model and Meaning

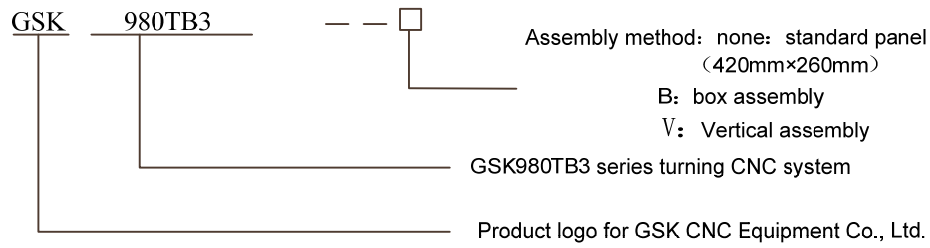


Fig. 1-3

### 1.4 Order

**GSK980TB3/TB3-V** can select Y as the additional axis and the user must remark it. Refer to the supplementary about Y axis explanation.

Table 1-4

| Model       | Explanation                                                            |
|-------------|------------------------------------------------------------------------|
| GSK980TB3   | 420mm×260mm aluminium operation panel                                  |
| GSK980TB3-V | 260mm×420mm Vertical aluminium operation panel                         |
| GSK980TB3-B | GSK980TB3<br>Be matched with AP01 to operation box (445mm×345mm×182mm) |

## PART TWO PROGRAMMING



## Chapter One Programming Fundamentals

### 1.1 Coordinate axis definition

It is important to stipulate the coordinate axis name and movement direction of CNC machine. Designers, operators and maintenance personnel of CNC machine should correctly understand it, otherwise, it causes the mistaken programming and data communication, operation accident, abnormal maintenance, and so on.

Fig.1-1-1 is a axis sketch map of CNC turning machine.

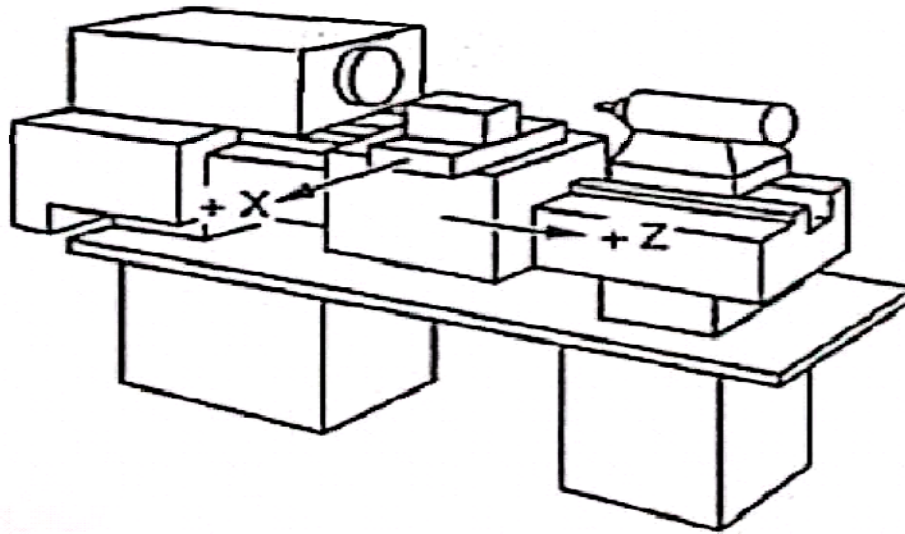


Fig.1-1-1

The system uses a rectangular coordinate system composed of X, Z axis to execute the positioning and the interpolation movement. X axis is in the direction of front and back in the plane, and Z axis is of left and right. The negative direction of them approach to the workpiece and positive one is away from it, which are shown in Fig.1-1-1.

The system supports a front tool post, a rear tool post function, and describes that the tool post before the workpiece is called as a front tool post and it behind the workpiece is called as a rear tool post. Fig. 1-1-2 is a coordinate system of the front tool post and Fig. 1—1-3 is a rear toolpost one. It shows exactly the opposite of X axes, but the same of Z axes from figures. In the manual, it will introduce programming application with the front tool post coordinate system in the following figures and examples.

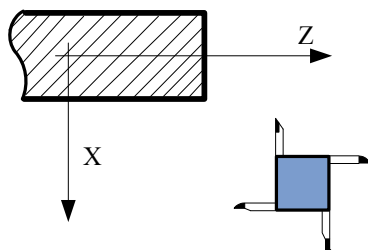


Fig.1-1-2 Front tool post coordinate system

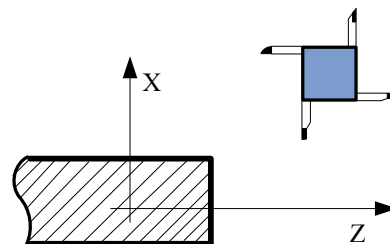


Fig.1-1-3 Rear tool post coordinate system

## 1.2 Machine coordinate system, Machine Zero

**Machine tool coordinate system** is a fixed one, its origin is taken as the machine zero installed on the max. travel in positive X, Z axis. The machine zero is defined after the machine is designed, manufactured and adjusted, and it is a fixed point. The machine zero is not defined when the CNC is turned on, and generally, the automatic or manual machine zero return is executed to create the machine coordinate system. CNC has created the machine coordinate system after the machine zero return is completed.

**Note:** Do not execute the machine zero function(such as G28) without the machine zero switch installed on the machine tool.

## 1.3 Workpiece Coordinate System and Reference Point(Program Zero)

A workpiece coordinate system (also called floating coordinate system) is used when programmers in programming, programmer selects a known point on the workpiece as reference point (also called program zero) to establish a new coordinate system, which is called a workpiece coordinate system. Once the workpiece coordinate system has been established, it is valid until it is replaced by a new one. When the system is turned off or power down, the program zero position is not saved. Using G50 for the system creates a workpiece coordinate system. When there is no G50 in programs, the current absolute coordinate value is taken the reference point to create a workpiece coordinate system. The reference point selection of the workpiece coordinate system should meet the simple programming, few dimension conversions and machining error. Generally, the reference point should be on the reference marked by the dimension or positioning reference. For turning machine programming, the reference point should be on the intersection point between the workpiece's axis and the end face of the chuck (Fig.1-3-1) or the workpiece's end face (Fig.1-3-2).

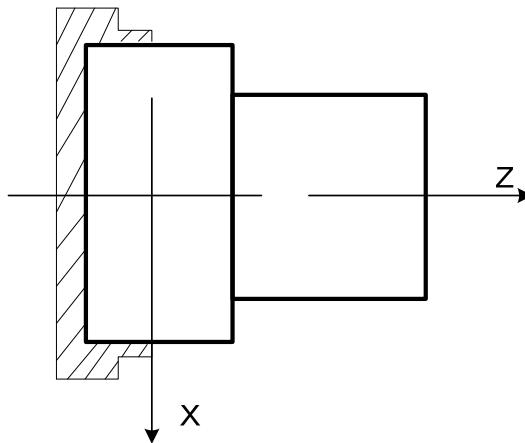


Fig.1-3-1 reference point on end face of chuck

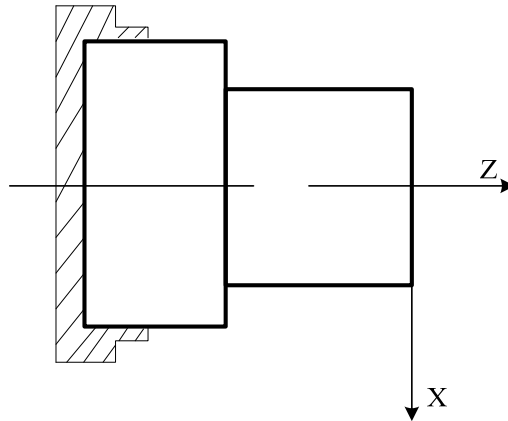


Fig. 1-3-2 reference point on workpiece's end face

#### 1.4 Controlled axes

Table 1-4-1

|                                   |                                |                  |
|-----------------------------------|--------------------------------|------------------|
| <b>GSK980TB3、<br/>GSK980TB3-V</b> | Controlled axes                | 3 axes X, Z, Y)  |
|                                   | Simultaneously controlled axes | 3 axes (X, Z, Y) |

#### 1.5 Input Increment

Table 1-5-1

| Input/output                | Least input increment  | Least command increment |
|-----------------------------|------------------------|-------------------------|
| Metric input /metric output | X: 0.001 mm (Diameter) | X: 0.0005 mm            |
|                             | Z: 0.001 mm            | Z: 0.001 mm             |
|                             | X: 0.001 mm (Radius)   | X: 0.001 mm             |
|                             | Z: 0.001 mm            | Z: 0.001 mm             |

Diameter/radius designation is set by NO:1#2 and the parameter is valid only to X.

The input increment is referred to the machine manufacture's User Manual.

#### 1.6 Maximum Stroke

Maximum stroke = least input increment×9999999

#### 1.7 Absolute Programming and Incremental Programming

The movement of the command axis is divided into: absolute command and incremental command. The absolute command is to use the end point of axis movement to execute programming, which is called absolute programming. The incremental command is to use the axis movement to directly execute programming, which is called incremental programming. For the system, the absolute programming uses X,Z and the incremental programming uses U, W.

Table 1-7-1

| Absolute command | Incremental command | Remark             |
|------------------|---------------------|--------------------|
| X                | U                   | X movement command |
| Z                | W                   | Z movement command |

Example: Using an absolute coordinates, incremental coordinates and compound coordinates compile A→B program described in Fig.1-7-1.

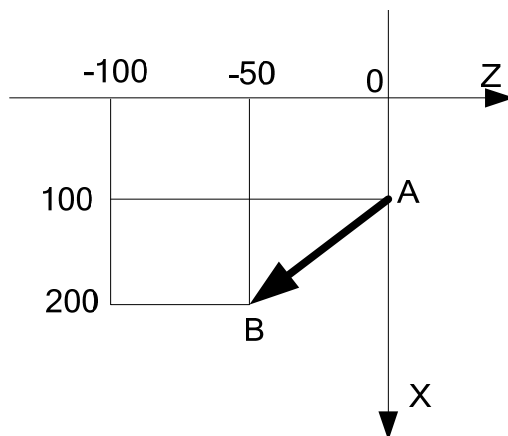


Fig. 1-7-1

Table 1-7-2

| Programming mode | Absolute programming                                                                             | Incremental programming | Compound programming |
|------------------|--------------------------------------------------------------------------------------------------|-------------------------|----------------------|
| Programming      | G1 X200 Z-50                                                                                     | G1 U100 W-50            | G1 X200 W-50         |
| Remark           | Suppose that current coordinate point of the tool is on A, the linear interpolation is executed. |                         |                      |

**Note:** When there are command address X/ U or Z/ W at the same time, #132 alarm occurs: X, U or Z,W exist simultaneously.

**Example:** G50 X10 Z20;  
G01 X20 W30 U20 Z30;

## 1.8 Diameter and Radius Programming

The appearance of the machined workpiece is a rotating body, among which X dimension can be specified by: the diameter and the radius, which is set by the bit of **NO:1#2**.

When **NO:1#2** is set to 1, the radius is specified to execute programming.

When **NO:1#2** is set to 0, the diameter is specified to execute programming.

Table 1-8-1 diameter, radius designation

| Item                                               | Diameter designation                                            | Radius designation   |
|----------------------------------------------------|-----------------------------------------------------------------|----------------------|
| Z command                                          | Not related to diameter, radius designation                     |                      |
| X command                                          | Diameter designation                                            | Radius designation   |
| Incremental command of address U                   | Diameter designation                                            | Radius designation   |
| Coordinate system setting (G50)                    | Diameter designation                                            | Radius designation   |
| X value of tool offset                             | NO:2#5 specifies the diameter designation or radius designation |                      |
| Radius command of circular interpolation (R, I, K) | Radius designation                                              | Radius designation   |
| X feedrate                                         | Radius change (mm/min, mm/r)                                    |                      |
| X position display                                 | Display diameter value                                          | Display radius value |

**Note 1:** The diameter designation is used except for special explanation in the User Manual.

**Note 2:** The tool offset using diameter/radius is defined that the outside diameter of workpiece uses diameter or radius when the tool offset is changed.

**Example:** when the diameter is specified, and the compensation value changes 10mm, the diameter value of the workpiece's outside diameter changes 10mm; when the radius is specified, and the compensation value changes 10mm, the diameter value of the workpiece's outside diameter changes 20mm.

## 1.9 Modal, Simple and Initial State

The modal is defined that after the function and state of the corresponding word are executed, they are valid till they are done again, namely, and the same functions and states are used in the following blocks are, the word need not be input again.

Example:

G0 X100 Z100; (rapid position to X100 Z100)

X120 Z30; (rapid position to X120 Z30, G0 is modal and can be omitted)

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

X100; (linear interpolation to X100 Z50, feedrate 300 mm/min, G1Z, 50, F300 are modal and can be omitted)

G0 X0 Z0; (rapid position to X0 Z0)

The simple is defined that after the function and state of the corresponding word are executed, they are valid one time, namely, and the same functions and states are used in the following blocks are, the word needs be input again.

The initial state is defined to the default function and state after the system is turned, namely, the system executes the initial function and state when the system is turned but does not define the corresponding function and state. The initial state of the system includes G00, G40, G97, G98, M05, M09, M33.

Example:

O0001;

G0 X100 Z100; (rapid position to X100 Z100, G0 is the system's initial state)

G1 X0 Z0 F100; (linear interpolation X0 Z0, feed per minute, feedrate 100 mm/min, G98 is the initial state after power on)



## Chapter Two Structure of an Part Program

A program is defined to a series collection of commands to control the CNC machine to complete workpiece machining. After the complied programs are input to the CNC system, the CNC system controls the tool movement along the linear and arc, the spindle starting/stopping, the cooling and the lubricating ON/OFF according to the commands. The commands in the program are compiled according to the actual movement sequence of the machine.

### 2.1 General Structure of a Program

A **program** consists of a sequence of block which is composed by words. Each block is separated by the block end command (ISO is LF, EIA is CR). “;” is used in the User Manual to mean the end of block.

General structure of a program is shown in Fig. 2-1-1.

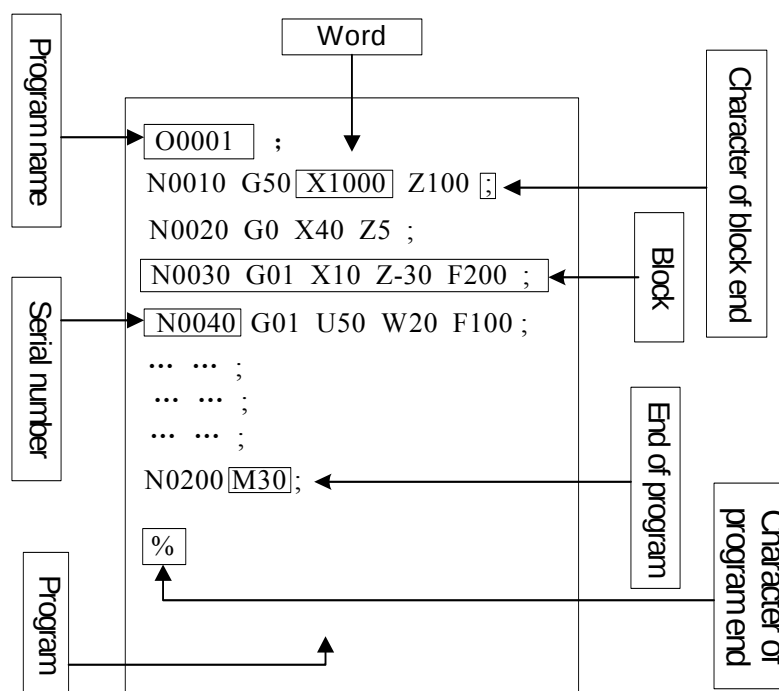
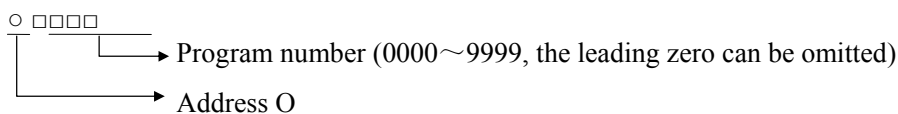


Fig. 2-1-1

#### 2.1.1 Program Name

There are most 500 programs stored in GSK980TDa. To identify it, each program has only one program name (there is no the same program name) beginning with command address O and the following 4 digits.



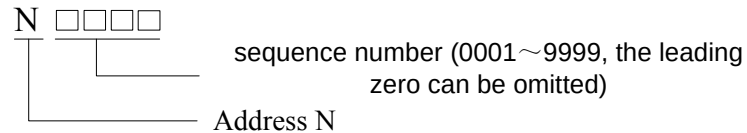
End of a program

A program starts with its program name and ends with “%”.

### 2.1.2 Sequence Number and Block

A program consists of many commands and one command unit is called a block (Fig.2-1-1). A block end command is used to separate blocks (Fig. 2-1-1), and “;” is the block end command in the User Manual.

The beginning of a block can use a sequence number composed by the address N and its following 4 digits.



The sequence of a sequence number is arbitrary (**NO:2 # 7** sets whether another sequence number is inserted) and its interval is not equal (the interval size is set by P50). All blocks can be with sequence numbers and some key blocks can be with them. Generally, sequence number is from the small to the big according to machining sequence.

### 2.1.3 Word

A word is an element of a block. A word is composed by an address and its following digit(some is with + or - before the digit), which is shown in Fig. 2-1-3:



Fig. 2-1-3-1

An address is one of English letters. The address describes the meaning of its following numerical. In the system, the useful addresses and meanings are shown in Fig. 2-1-3-1. Some address has different meanings according to different commands.

Table 2-1-3-1 address list

| Address | Value range             | Function             |
|---------|-------------------------|----------------------|
| O       | 0~9999                  | Program name         |
| N       | 1~9999                  | Block number         |
| G       | 00~99                   | Preparatory function |
| X       | -9999.999~9999.999 (mm) | X coordinate         |
|         | 0~9999.999 (s)          | Pause time           |
| Z       | -9999.999~9999.999 (mm) | Z coordinate         |
| U       | -9999.999~9999.999 (mm) | X increment          |

| Address | Value range                      | Function                                                              |
|---------|----------------------------------|-----------------------------------------------------------------------|
|         | -9999.999~9999.999 (mm)          | X finishing allowance in G71,G72, G73                                 |
|         | 0.001~9999.999 (mm)              | Cutting depth in G71                                                  |
|         | -9999.999~9999.999 (mm)          | Travel of X tool retraction in G73                                    |
| W       | -9999.999~9999.999 (mm)          | Z increment                                                           |
|         | 0.001~9999.999 (mm)              | Cutting depth in G72                                                  |
|         | -9999.999~9999.999 (mm)          | Z finishing allowance in G71,G72, G73                                 |
|         | -9999.999~9999.999 (mm)          | Z tool retraction in G73                                              |
| R       | 0~9999.999 (mm)                  | Arc radius                                                            |
|         | 0.001~9999.999 (mm)              | Tool retraction in G71, G72                                           |
|         | 1~9999999 (times)                | Roughing cycle times in G 73                                          |
|         | 0~9999.999 (mm)                  | Tool retraction in G74, G75                                           |
|         | 0~9999.999 (mm)                  | Tool retraction distance G74, G75                                     |
|         | 0~9999.999 (mm)                  | Finishing allowance in G76                                            |
|         | -9999.999~9999.999 (mm)          | Taper in G90, G92, G94, G96                                           |
| I       | -9999.999~9999.999 (mm)          | X vector between arc center and starting point                        |
|         | 0.06~25400 (tooth/inch)          | Metric thread tooth                                                   |
| K       | -9999.999~9999.999 (mm)          | Z vector between arc center and starting point                        |
| F       | 1~8000 (mm/min)                  | Feedrate per minute                                                   |
|         | 0.001~500(mm/r)                  | Feedrate per rev                                                      |
|         | 0.001~500 (mm)                   | Metric thread lead                                                    |
| S       | 0~9999 (r/min)                   | Specifying spindle speed                                              |
|         | 0~9999 (m/min)                   | Specifying spindle constant surface speed                             |
|         | 00~04                            | Multi-gear spindle output                                             |
|         | 10~99                            | Subprogram call                                                       |
| T       | 0100~0800                        | Tool function, subprogram call                                        |
| M       | 00~99                            | Miscellaneous function output, program executed flow, subprogram call |
| P       | 1~9999999 (0.001s)               | Pause time                                                            |
|         | 0~9999                           | Call subprogram number                                                |
|         | 0~999                            | Call times of subprogram                                              |
|         | 0.001~9999.999 (mm)              | X circular moving distance in G74, G75                                |
|         | See Chapter 3.4, G76 Explanation | Thread cutting parameter in G76                                       |
|         | 1~9999                           | Initial block number of finishing in the compound cycle command       |
| Q       | 1~9999                           | End block number of finishing in the compound cycle                   |
|         | 0.001~9999.999 (mm)              | Z circular moving distance in G74, G75                                |
|         | 1~9999.999 (mm)                  | First cutting depth in G76                                            |

| Address | Value range | Function            |
|---------|-------------|---------------------|
| H       | 01 ~ 99     | Operand in G65      |
| L       | 01 ~ 99     | Thread heads in G92 |

The limited values described in Table 2-1-3-1 are for the CNC device, but the limited values for the machine are not described here. Please refer to the user manual, another user manual from the machine manufacturer when programming.

## 2.2 Relationship between Command Numerical Value and Decimal Point

In the system, some command cannot be with a decimal point, and NO:11#0 sets whether the decimal point is used when programming, and the relationship between the command numerical value and the decimal point is shown in Fig.2-2-1:

**Table 2-2-1**

| Address | Having a decimal point | NO:11#0=1                                     | NO:11#0=0                                       | Remark                                 |
|---------|------------------------|-----------------------------------------------|-------------------------------------------------|----------------------------------------|
| X       | Yes                    | Command: G1 X20<br>Positioning point: 20      | Command: G1 X20<br>Positioning point: 0.02      |                                        |
|         |                        | Command: G4 X20<br>Delay: 20s                 | Command: G4 X20<br>Delay: 0.02s                 |                                        |
| Z       | Yes                    | Command: G1 Z20<br>Positioning point: 20      | Command: G1 Z20<br>Positioning point: 0.02      |                                        |
| U       | Yes                    | Command: G1 U20<br>Incremental value:<br>20mm | Command: G1 U20<br>Incremental value:<br>0.02mm |                                        |
| W       | Yes                    | Command: G1 W20<br>Incremental value:<br>20mm | Command: G1 W20<br>Incremental value:<br>0.02mm |                                        |
| R       | Yes                    |                                               |                                                 | It is the same as the address X        |
| I       | Yes                    |                                               |                                                 |                                        |
| K       | Yes                    |                                               |                                                 |                                        |
| P       | No                     | Command: G4 P2<br>Delay: 0.002s               |                                                 | It is not related to the decimal point |
| S       | No                     |                                               |                                                 |                                        |

## 2.3 Subprogram

### 2.3.1 Main Program and Subprogram

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

run separately. The system returns to the main program to continue when the subprogram ends, which is shown below:

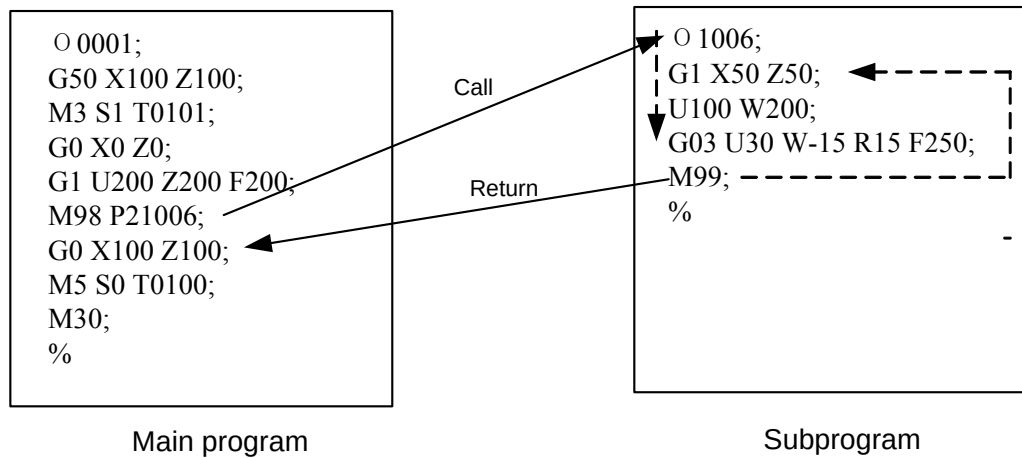
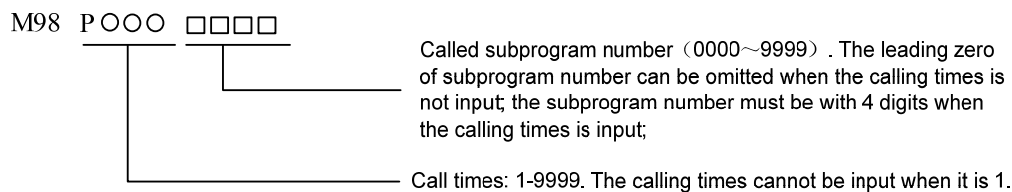


Fig. 2-3-1-1

### 2.3.2 Subprogram Call (M98)

**Command format:**

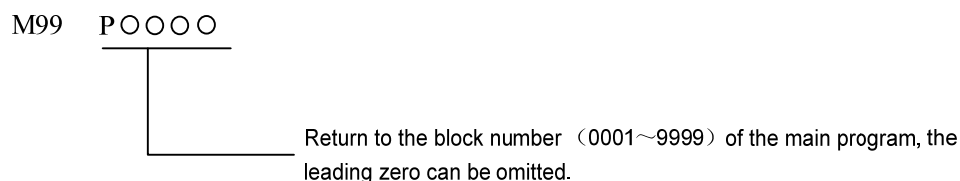


**Command function:** After other commands of current block are executed in M98, CNC calls subprograms specified by P instead of the next block, and subprograms are executed 9999 times at most.

**Note:** The system cannot call a subprogram in MDI mode.

### 2.3.3 Return from subprogram (M99)

**Command format**



**Command function:** After other commands of the current block (in the subprogram) are executed, the system returns to the main program and continues to execute next block specified by P, and calls a block following M98 of current subprogram when P is not input. The current program is executed repeatedly when M99 is defined to end of the main program.

Example: Execution path of calling subprogram (with P in M99) is shown in Fig. 2-3-3-1.

Execution path of calling subprogram (without P in M99) is shown in Fig. 2-3-3-2.

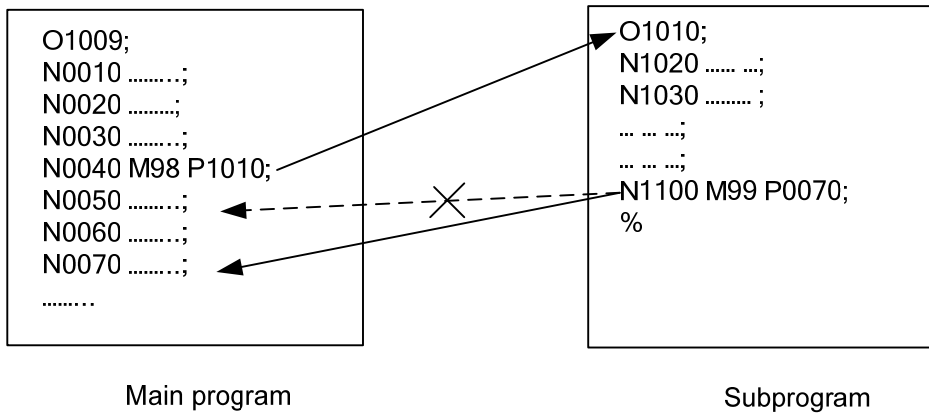


Fig. 2-3-3-1

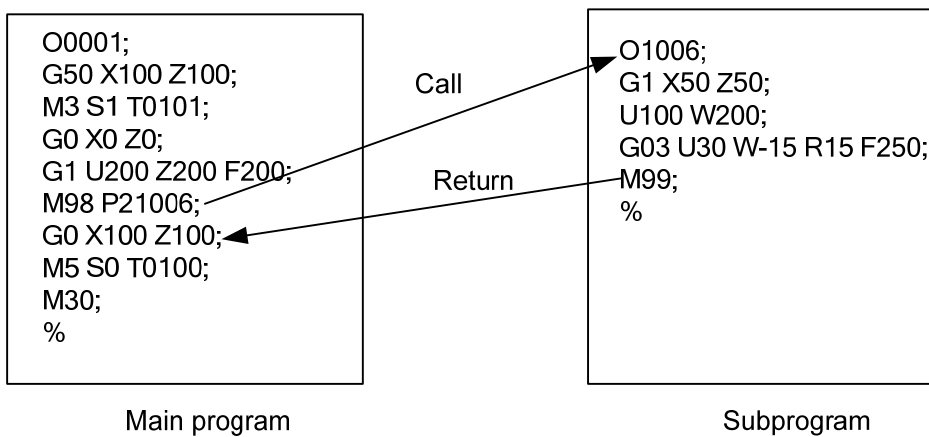


Fig. 2-3-3-2

The system can call fourfold-embedded subprograms, namely can call other subprograms in another subprogram (Fig.2-3-3-3 is an example of double-embedded).

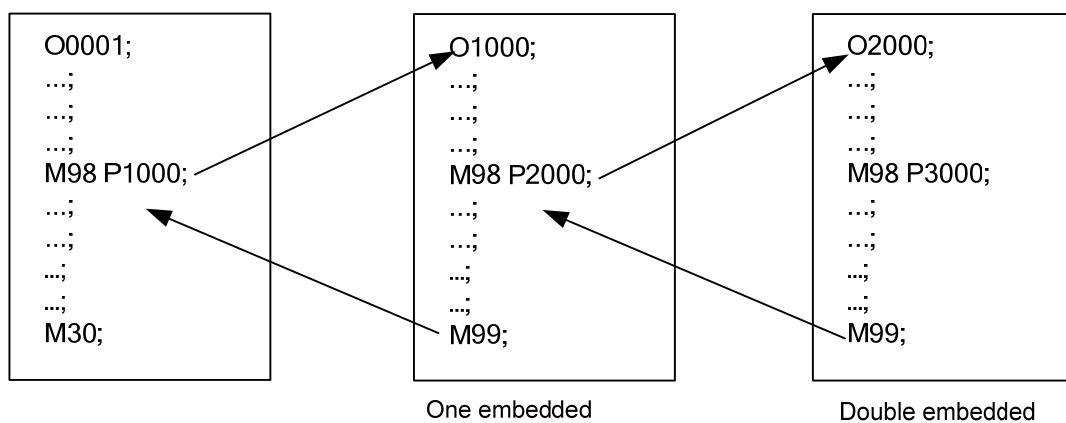
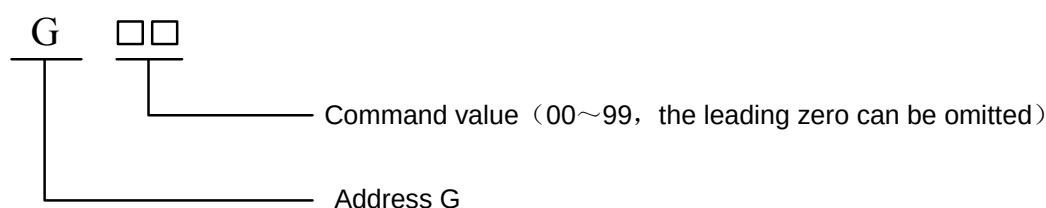


Fig. 2-3-3-3 double-embedded subprogram

## Chapter Three Preparatory Function G Code

### 3.1 Category of Preparatory Function G Code

Preparatory function---A G Code consists of command address G and its following 2 digits numerical value, and is used for defining the motion path of the tool relative to the workpiece, setting the coordinates and so on. G Codes are shown in Fig. 3-1-1.



G words are divided into 6 groups (00, 01, 02, 03, 06, 07). That commands in the group 01 are simple and others are modal.

After G Codes are executed, their defined functions and states are valid until they are changed by others in the same group, the previous functions and states are cancelled.

The initial G Code is the initial mode after the system is turned on, the commands in the initial mode include G00, G97, G98, G40, G21.

The defined functions and state are valid one time after the simple G Code is executed, and it must be input again when it is executed every time.

After the system is switched on, the valid modal G Codes which are not executed their functions or states are called initial mode G command. Take it as the initial mode G Code to be executed when it is not be input after the system is switched on. The initial commands of GSK980TDa include G00, G40, G97, G98.

Several G codes (Group 00 and 01 must not be in the same block) which belong to different groups can be commanded in the same block. No.129 alarm occurs when more than two G codes which belong to the same group are commanded in the same block. When G codes with common word which belong to different group are in the same block, their functions are valid simultaneously and are unconcerned with their sequence. The system alarms when other G codes except for ones described in Table 3-1-1 or G codes which have no selection functions.

Table 3-1-1 G Code list

| Command | Group | Format                      | Explanation                                                            |
|---------|-------|-----------------------------|------------------------------------------------------------------------|
| *G00    | 01    | G00 X(U)___Z(W)___          | Positioning, rapid traverse, speed rate of each axis set by parameters |
| G01     |       | G01 X(U)___Z(W)___F         | Linear interpolation                                                   |
| G02     |       | G02 X(U)___Z(W)___R(I_K_) F | Circular interpolation CW                                              |
| G03     |       | G03 X(U)___Z(W)___R(I_K_) F | Circular interpolation CCW,                                            |
| G04     | 00    | G04 P_; or G04 X_;          | Dwell                                                                  |

|      |    |                                                                       |                                                                |
|------|----|-----------------------------------------------------------------------|----------------------------------------------------------------|
| G10  | 00 | G10 P__ (parameter number) Q__ (numerical value)                      | A program specifying a parameter function                      |
| G20  | 06 | G20                                                                   | Inch selection                                                 |
| *G21 |    | G21                                                                   | Metric selection                                               |
| G28  | 00 | G28 X(U)___Z(W)___                                                    | Return to reference point, and X, Z specifies the middle point |
| G31  | 00 | G31 X (U)___Z (W)___F___                                              | Skip function                                                  |
| G32  | 01 | G32 X(U)___Z(W)___F(I)                                                | Invariable pitch thread cutting                                |
| G33  | 01 | G33 Z(W)___F(I); G33 X(U)___F(I)                                      | Tapping cycle                                                  |
| G34  | 01 | G34 X(U)___Z(W)___F(I) K___                                           | Variable pitch thread cutting                                  |
| *G40 | 07 | G40                                                                   | Tool radius compensation cancel                                |
| G41  |    | G41                                                                   | Tool radius compensation (left)                                |
| G42  |    | G42                                                                   | Tool radius compensation (right)                               |
| G50  | 00 | G50 X(U)___Z(W)___                                                    | Coordinate system setting                                      |
| G51  | 00 | G51 X (U)___Z (W)___                                                  | Local coordinate system function                               |
| G65  | 00 | G65 Hm P#I Q#J R#K                                                    | Macro code (see the following)                                 |
| G70  | 00 | G70 P(ns) Q(nf)                                                       | Finishing cycle                                                |
| G71  |    | G71 U(ΔD) R(E)<br>G71 P(NS) Q(NF) U (ΔU) W(ΔW) F(F) S(S) T(T)         | Outer roughing cycle                                           |
| G72  |    | G72 W (ΔD) R (E)<br>G72 P(NS) Q(NF) U(ΔU) W(ΔW) F(F) S(S) T(T)        | End roughing cycle                                             |
| G73  |    | G73 U (ΔI) W (ΔK) R (D)<br>G73 P(NS) Q(NF) U(ΔU) W(ΔW) F(F) S(S) T(T) | Closed cutting cycle                                           |
| G74  |    | G74 R(e)<br>G74 X(U) Z(W) P(Δi) Q(Δk) R(Δd) F(f)                      | End deep hole machining cycle                                  |
| G75  |    | G75 R(e)<br>G75 X(U) Z(W) P(Δi) Q(Δk) R(Δd) F(f)                      | Outer/inner grooving cycle                                     |
| G76  |    | G76 P(m) (r) (a) Q(Δdmin) R(d)<br>G76 X(U) Z(W) R(i) P(k) Q(Δd) F(L)  | Compound thread cutting cycle                                  |
| G90  | 01 | G90 X(U)___Z(W)___R___F___                                            | Outer, inner turning cycle                                     |
| G92  |    | G92 X(U)___Z(W)___R___F(I)___J___K___                                 | Thread turning cycle                                           |
| G94  |    | G94 X(U)___Z(W)___R___F___                                            | End turning cycle                                              |
| G96  | 02 | G96 S                                                                 | Constant surface control                                       |
| *G97 |    | G97 S                                                                 | Constant surface control cancel                                |
| *G98 | 03 | G98                                                                   | Feed per minute                                                |
| G99  |    | G99                                                                   | Feed per rev                                                   |

**Note 1:** When the system is turned on, it is in the state of G Code with \*.

**Note 2:** G Codes in Group 00 are simple.

## 3.2 Simple G Code

### 3.2.1 Rapid Positioning (G00)

**Command format:** G00 X (U) \_ Z (W) \_ ;

**Function:** X, Z rapidly traverses at the respective traverse speed to the position specified by X(U), Z(W).

**Explanation:** X (U): absolute (incremental) coordinate of X positioning end point;

Z (W): absolute (incremental) coordinate of Z positioning end point;

1. X, Z rapidly traverses at the respective traverse speed and their combined path is really not linear, so maybe they cannot reach the end points simultaneously, please pay attention to it when programming ( Fig. 3-2-1-1).

2. X, Z rapidly traverses at the speed separately set by P21, P22 are adjusted by pressing  on the operation panel, or are performed by selecting manual rapid override



which has five grades including Fo, 25%, 50%, 75%, 100%. (Fo speed is set by P32)

3. The tool does not traverse when there is a positioning parameter after G00, the system only changes the current tool traverse mode into G00.

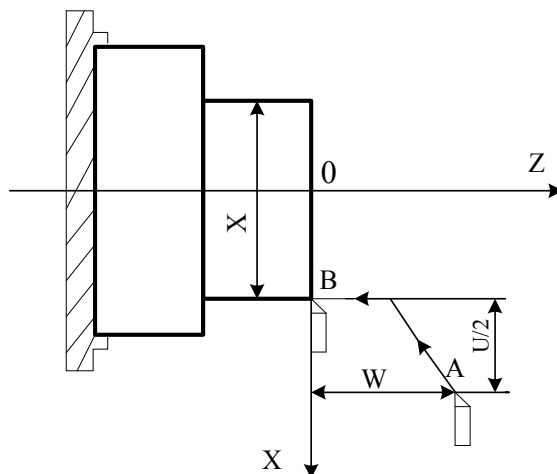


Fig. 3-2-1-1

**Example:** the tool rapidly traverses from A to B, which is shown in Fig. 3-2-1-2:

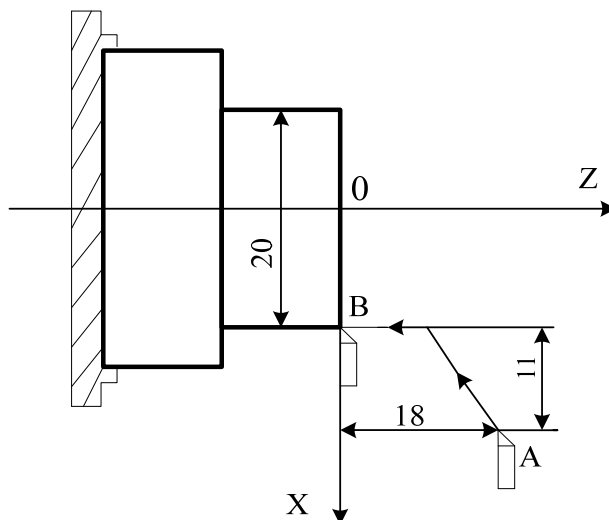


Fig. 3-2-1-2

Programming:

G0 X20 Z0; (absolute programming, diameter programming)

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

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

### 3.2.2 Linear Interpolation (G01)

**Command format:** G01 X (U) \_ Z (W) \_ F\_;

**Function:** the tool traverses to the specified position at the feedrate (mm/min) specified by F. The interpolation path is shown in Fig. 3-2-2-1.

**Explanation:** X (U): absolute (incremental) coordinate of X interpolation's end point;

Z (W): absolute (incremental) coordinate of Z interpolation's end point;

F: is combined feedrate of X, Z, is modal. Its value is related to G98 OR G99, which is shown in 3-2-2-1:

Table 3-2-2-1

|             | G98 (mm/min) | G99 (mm/r) |
|-------------|--------------|------------|
| Value range | 1~8000       | 0.001~500  |

Command path:

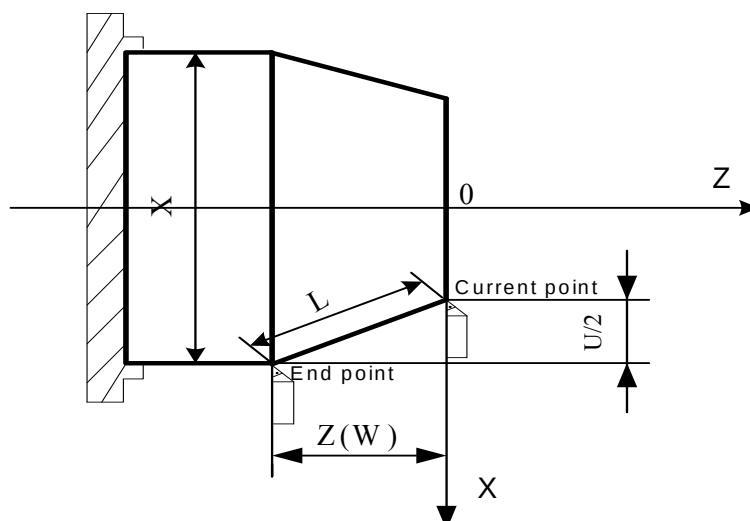


Fig. 3-2-2-1

1. Feedrate specified by F is valid until a new F value is specified. Feedrate specified by F is counted by linear interpolation. When F is not commanded in programs, feedrate uses F feedrate of initial speed. (refer to P103 about setting) .

For interpolation mode of two axes simultaneously moving, F is specified to the composite feedrate of two axes.

It shows from Fig. 3-2-2-1:

$$X \text{ feedrate } F_X = \frac{U}{L} \times F :$$

$$Z \text{ feedrate } F_Z = \frac{W}{L} \times F \quad (L = \sqrt{U^2/4 + W^2}, \text{ U is a diameter value})$$

2. P27 can set the upper of cutting feedrate F. When the actual cutting feedrate (feedrate after using override) exceeds the upper, NO.11 alarm occurs, feedrate unit is mm/min.
3. When the position in G Code following G01 is not commanded, the tool does not traverse, the system only changes the current tool traversing mode to G01.



4. Pressing the buttons can tune the feedrate override, which is divided into 16 grades from 0%~150%.

Example: compile the linear interpolation program from current point to end point, which shown in

Fig. 3-2-2-2.

Program (diameter programming):

G01 X60.0 Z-25; (absolute programming)

G01 U20.0 W-25.0; (incremental programming)

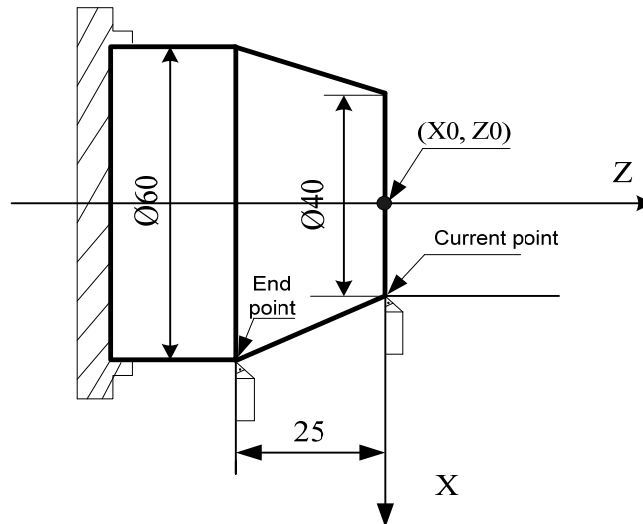


Fig. 3-2-2-2

### 3.2.3 Circular Interpolation (G02/G03)

Command format:

$$\left\{ \begin{array}{c} G02 \\ G03 \end{array} \right\} X(U) \_ Z(W) \_ \left\{ \begin{array}{c} R\_ \\ I\_K\_ \end{array} \right\} F\_;$$

**Function:** X, Z from starting point (the position before the current block runs) moves along the radius specified by R or CW/CCW interpolates along the circle center defined by I, K value to end point specified by X(U), Z(W).

G02 motion path is an arc from starting point to end point along CW (the rear post tool coordinate system)/CCW (the front tool post coordinate system), which is shown in Fig.3-2-3-1.

G03 motion path is an arc from starting point to end point along CCW (the rear post tool coordinate system)/CW (the front tool post coordinate system), which is shown in Fig.3-2-3-2.

**Explanation:** X (U): absolute (incremental) coordinate of X circular interpolation's end point;

Z (W): absolute (incremental) coordinate of Z circular interpolation's end point;

R: arc radius;

I: X difference value of circle center relative to starting point of arc; (radius command)

K: Z difference value of circle center relative to starting point of arc;

X, U, Z, W, I, K range: -9999.999 mm~9999.999 mm, R range: 0mm~9999.999mm

F: cutting speed of arc.

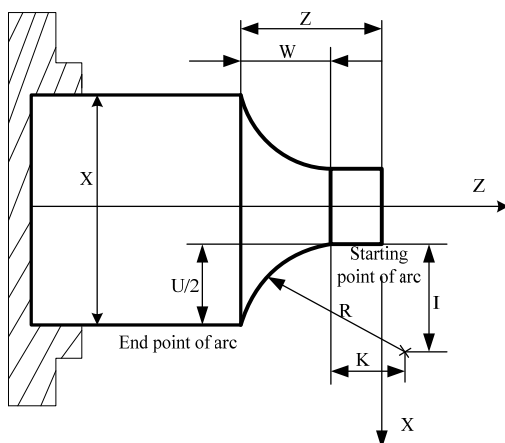
**Command path:**

Fig.3-2-3-1 G02 path

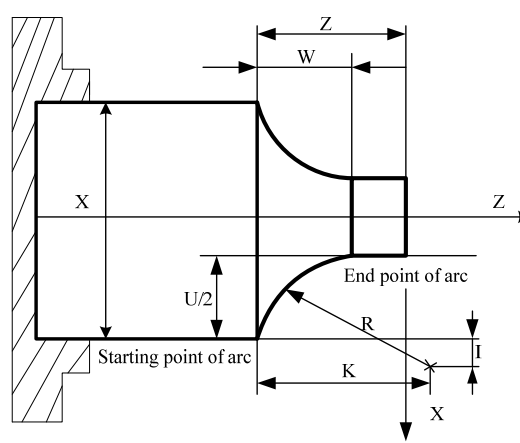


Fig.3-2-3-2 G03 path

CW or CCW interpolation is related that the system uses the front tool post coordinate system or the rear tool post coordinate system. The system uses the front tool post coordinate system, which is shown in Fig. 3-2-3-3.

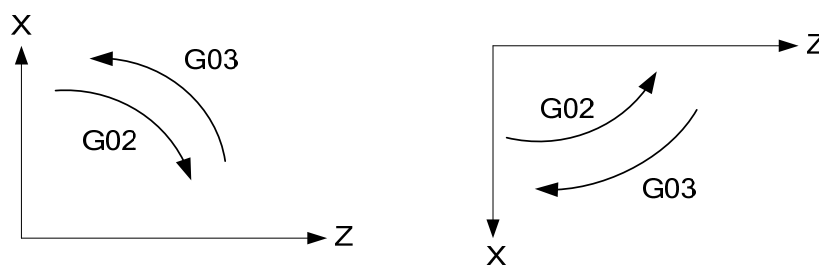


Fig. 3-2-3-3

Example: G02 compiles a program shown in Fig. 3-2-3-4:

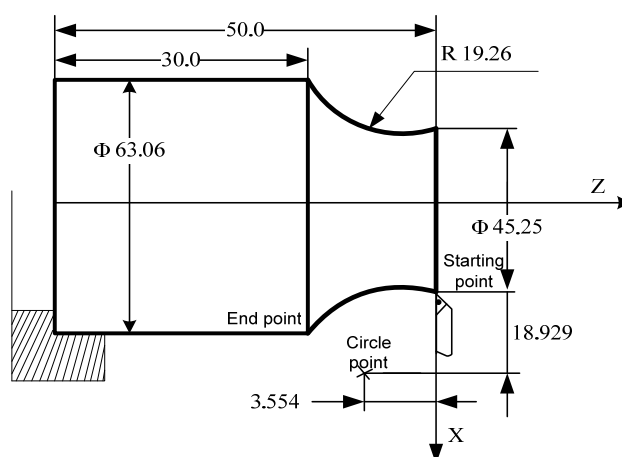


Fig. 3-2-3-4

Program (the current point of the tool is the starting point):

G02 X63.06 Z-20 R19.26 F300 ;                      or  
G02 U17.81 W-20.0 R19.26 F300 ;                      or

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

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

**Note 1:** the arc center is specified by I, K which separately corresponds to X, Z (Fig. 3-2-3-1, Fig.3-2-3-2). I, K mean the vector component from starting point to circle center, with signs.

**Note 2:** when I, K or, R is not specified in the command format, the system executes the interpolation along G01 path.

**Note 3:** one or all of X(U), Z (W) can be omitted. Omitting one means the starting point and end point of the axis are consistent; omitting all means the starting point and end point are in the same position.

**Note 4:** when X (U), Z (W) are omitted simultaneously, and I, K are circle center, which means the arc path is a full circle; I, K and R cannot be command simultaneously, otherwise, the system alarms.

**Note 5:** when I=0 or K=0, they can be omitted.

**Note 6:** when R is specified, R>0, the specified arc is less than 180 degree; when the arc is more than 180 degree, I, K are specified; R, I, K are simple.

**Note 7:** when R error range exceeds the value set by No. 98, No.26 alarm occurs.

**Note 8:** in MDI mode, I, K are valid but I, K values are not displayed.

### 3.2.4 Chamfering/Cornering Arc

Command format: **C\_ : chamfer**

**B\_ : chamfer arc transition**

**Function:** the above is added to the end of the block in G01 or G02, G03, the chamfering at the cornering or the arc transition is executed automatically.

**Explanation:** the chamfering following C specifies the distance from the virtual inflection point to corner starting point and end point, the virtual inflection point is presumed not to execute the corner, the corner point actually exists, which is shown below:

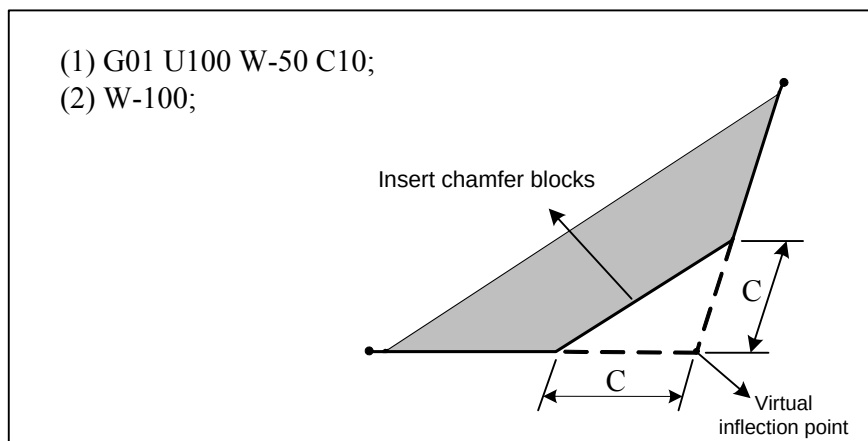


Fig. 3-2-4-1

After the chamfer arc transition is B, the chamfer arc radius is specified, which is shown below:

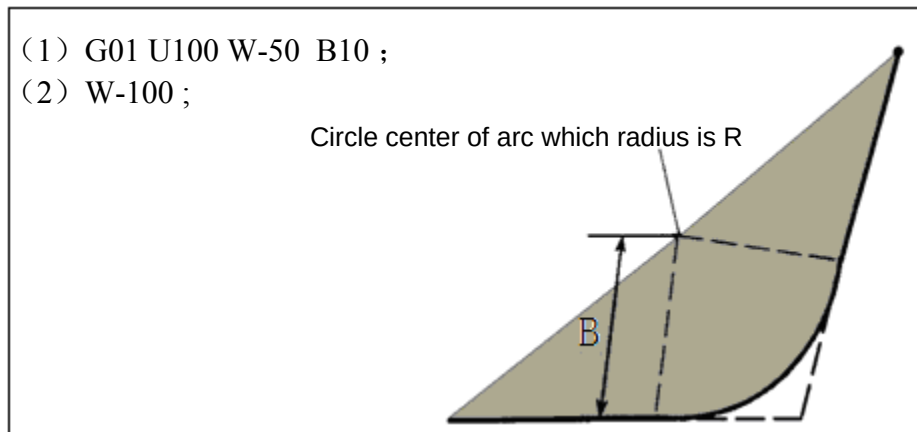


Fig. 3-2-4-2

**Limits:**

1. Cornering and chamfering arc are executed only in the specified plane, and cannot be executed by the parallel axes.
2. After a block for cornering and chamfering arc transition, G01 or G02, G03 must be followed, and the cornering and chamfering arc transition are disabled when there is no these commands in the following block.
3. The system alarms when the inserted block for the chamfering or arc transition causes the tool exceeding the previous interpolation movement range.
4. The Chamfering or cornering arc transition cannot be specified in the block where the coordinate system is changed or which is followed by the reference point return is executed.
5. The chamfer arc transition cannot be specified in the block used for the thread machining.
6. The chamfering and cornering values cannot be negative.

**3.2.5 Dwell G04**

**Command format:** G04 P\_\_ ; or

G04 X\_\_ ;

G04 U\_\_ ;

**Function:** When G04 executes the dwell operation, the system delays the specified time to execute the next block. The specified time range: 0.001s~9999.999s.

When P, X or U is specified, it means the exact stop between block, dwell time is influenced by No. 031, the bigger the parameter set value, the longer the dwell time is;

**Explanation:** when No:11#0 is set to 1, the unit is shown in Fig. 3-2-5-1:

Table 3-2-5-1

| Address | P      | X  | U  |
|---------|--------|----|----|
| Unit    | 0.001s | 1s | 1s |

When No:11#0 is set to 0, the unit is shown in Fig. 3-2-5-2:

| Address | P      | X      | U      |
|---------|--------|--------|--------|
| Unit    | 0.001s | 0.001s | 0.001s |

Note 1: when G04 is execute, P and X or P and U are in the same block, X, U are valid, P is invalid.

Note 2: G04 and M98 cannot be in the same block, otherwise, the alarm (eer76) occurs.

Note 3: G04 and other commands in the Group 01 cannot be in the same block, otherwise, the alarm (err131) occurs.

### 3.2.6 Programs Specifying Parameter Function (G10)

**Command format:** G10 P (parameter number) Q (numerical value)

**Function:** When G10 is specified, the actual output of related parameters from the one specified by G10 is executed

Example: G10 P45 Q1000

After the block is executed, No. 45 value is automatically changed into 1000

**Explanation:** (1) Parameter range specified by P: all data parameter of the system;

(2) Q range and unit are related the specified parameter;

(3) Parameter (Q) set value has no decimal point. It automatically cleared away by the system when the decimal point is input when programming.

### 3.2.7 Machine Zero Return (G28)

**Command format:** G28 X (U) \_ Z (W) \_ ;

**Function:** The commanded axis goes through the middle point specified by X (U), Z(W) to return to the machine zero. One or two axes can be specified in G28.

Table 3-2-7-1

| Command           | Function                                       |
|-------------------|------------------------------------------------|
| G28 X (U)         | X returns to machine zero, Z remains unchanged |
| G28 Z (W)         | Z returns to machine zero, X remains unchanged |
| G28               | X, Z remain unchanged                          |
| G28 X (U) _ Z (W) | X, Z return to machine zero simultaneously     |

**Explanation:** G28 operation process (Fig3-2-7-1):

- (1) Rapid traverse to middle point from current position(A point→B point) ;
- (2) Rapid traverse to reference point from the middle point(B point→R point) ;
- (3) When the machine is not locked and after the machine reference point return is completed, the machine zero return indicator is ON and the machine coordinates are cleared.

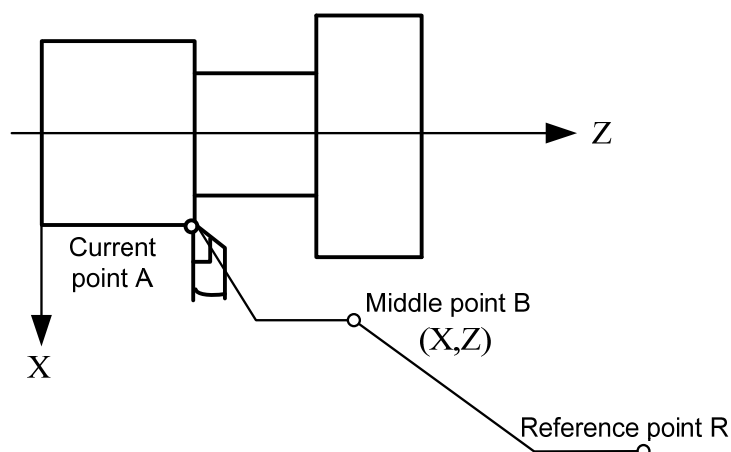


Fig. 3-2-7-1

**Note 1:** After the system is turned on, the system has not executed the manual machine zero return firstly, the path from the middle point to the machine zero is the same that of the manual machine zero return when G28 is executed.

**Note 2:** A point → B point and B point → R point, X, Z moves at its respective speed, so, their paths are not linear.

**Note 3:** The machine zero return function cannot be executed when there is no the machine zero installed on the machine.

**Note 4:** When No.006: Bit6 is set to 1, the system is firstly turned on to execute G28 and after the machine zero return is executed, the set coordinate system is valid; when G28 is executed again and after the machine zero return is executed, the set coordinate system is invalid.

### 3.2.8 Workpiece Coordinate System Setting (G50)

#### 1) Workpiece coordinate system setting

**Command format:** G50 X\_ Z\_ ;

**Function:** Set a workpiece coordinate system. Two code parameters specify the absolute coordinate value of tool nose position of current tool post on a new workpiece coordinate system. G50 does not make the motion axis move.

**Explanation:** X : X absolute coordinate of current tool nose in the workpiece coordinate system;

Z : Z absolute coordinate of current tool nose in the workpiece coordinate system;

1. After the coordinate system is established, the position of the absolute command is presented in the coordinate system till one new coordinate system is established by G50.
2. When the diameter programming is specified, the diameter is specified to X; when the radius programming is specified, the radius is specified to.

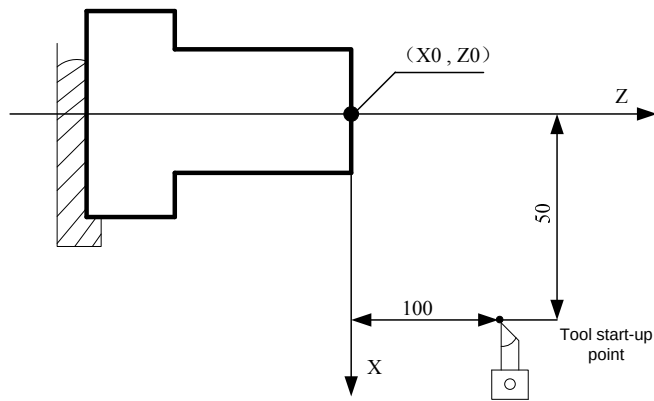


Fig. 3-2-8-1

After G50 X100 Z100 is executed, the workpiece coordinate system is thus set (shown in Fig. 3-2-8-1) and the tool nose's current position in the workpiece coordinate system is defined. Refer to Section 4.4 Tool Setting for coordinates setting methods.

**Note 1:** G50 cannot be in the same block with the commands included to Group 01, otherwise, ERR131 alarm occurs.

**Note 2:** When using G50 sets the coordinate system in the tool offset cancel mode. After the coordinate is set, the absolute coordinates are those of G50 set value; the tool offset cancel is executed in MDI mode: "T0100 G00 U0 W0". Suppose that the current tool offset state is T0101.

**Note 3:** When using 50 sets the coordinate system in tool offset state, there are two kind of displaying absolute coordinates:

**A.** The tool offset has been executed(have movement command after tool offset) and after the setting is done, the absolute coordinates are the one set by G50, which is shown below:

Table 3-2-8-1

| Program(execute tool compensation in coordinate offset mode) | Absolute coordinate display value | No. 01 tool compensation value |
|--------------------------------------------------------------|-----------------------------------|--------------------------------|
| G0 X0 Z0                                                     | X: 0 Z: 0                         | X: -12<br>Z: -23               |
| T0101                                                        | X: 12 Z: 23                       |                                |
| G0 X0 Z0                                                     | X: 0 Z: 0                         |                                |
| G50 X20 Z20                                                  | X: 20 Z: 20                       |                                |

**B.** The tool offset has not executed(have no movement commands after tool offset), including tool offset cancel and tool offset setting, after the setting is done, the absolute coordinates are the tool offset values, which is shown below:

Table 3-2-8-2

| Program(execute tool compensation in coordinate offset mode) | Absolute coordinate display value | No. 01 tool compensation value |
|--------------------------------------------------------------|-----------------------------------|--------------------------------|
| G0 X0 Z0                                                     | X: 0 Z: 0                         | X: -12<br>Z: -23               |
| T0101                                                        | X: 12 Z: 23                       |                                |
| G0 X50 Z50                                                   | X: 50 Z: 50                       |                                |
| T0100                                                        | X: 38 Z: 27                       |                                |
| G50 X20 Z20                                                  | X: 8 Z: -3                        |                                |

Table 3-2-8-3

| Program(execute tool compensation in coordinate offset mode) | Absolute coordinate display value | No. 01 tool compensation value |
|--------------------------------------------------------------|-----------------------------------|--------------------------------|
| G0 X0 Z0                                                     | X: 0 Z: 0                         | X: -12<br>Z: -23               |
| T0101                                                        | X: 12 Z: 23                       |                                |
| G50 X20 Z20                                                  | X: 32 Z: 43                       |                                |

## 2) Coordinate system translation

**Command format:** G50 U\_ W\_ ;

**Function:** The tool nose position in the previous absolute coordinate system translates the distance set by a parameter. Namely, the tool nose position in the new coordinate system corresponding to the previous absolute coordinate system is X+U, Z+W.

**Explanation:** When the diameter programming is specified, X is done by diameter; when the radius programming is specified, X is done by radius.

**Note:** There is no M, S, T in the block of G50, otherwise, the system alarms.

### 3.2.9 Workpiece Coordinate System Offset (G51)

**Command format:** G51 X (U) Z (W)

**Function:** G51 is specified to complete the workpiece coordinate system offset when programming.

**Explanation:** When the absolute coordinate programming is used, for example, G51 X\_ Z\_ , means that the current coordinate system offsets to the position specified by X, Z;

When the incremental coordinate programming is used, for example, G51 U\_ W\_ , means the current coordinate system offsets the displacement values specified by U, W;

Specify G51, X0, Z0 to cancel the workpiece coordinate offset and recover the previous workpiece coordinate system.

**Example 1:** use “G51 X\_ Z\_ ” to execute the offset

| Program       | Absolute coordinates | Machine coordinates | Explanations                                                                                                                |
|---------------|----------------------|---------------------|-----------------------------------------------------------------------------------------------------------------------------|
| G50 X100 Z100 | X100 Z100            | X100 Z100           | G50 sets coordinate system                                                                                                  |
| G51 X-10 Z-10 | X110 Z110            | X100 Z100           | G51 executes the offset corresponding to the previous workpiece coordinate system instead of the current coordinate system. |
| G51 X20 Z20   | X80 Z80              | X100 Z100           |                                                                                                                             |
| G0 X10 Z10    | X10 Z10              | X30 Z30             |                                                                                                                             |
| G51 X0 Z0     | X30 Z30              | X30 Z30             | Cancel the workpiece coordinate system offset                                                                               |

**Example 2:** use “G51 U\_ W\_ ” to execute the offset.

| Program       | Absolute coordinates | Machine coordinates | Explanations               |
|---------------|----------------------|---------------------|----------------------------|
| G50 X100 Z100 | X100 Z100            | X100 Z100           | G50 sets coordinate system |

|               |           |           |                                                                                                                             |
|---------------|-----------|-----------|-----------------------------------------------------------------------------------------------------------------------------|
| G51 U-10 W-10 | X110 Z110 | X100 Z100 | G51 executes the offset corresponding to the previous workpiece coordinate system instead of the current coordinate system. |
| G51 U20 W20   | X90 Z90   | X100 Z100 |                                                                                                                             |
| G0 X10 Z10    | X10 Z10   | X20 Z20   |                                                                                                                             |
| G51 X0 Z0     | X20 Z20   | X20 Z20   | Cancel the workpiece coordinate system offset                                                                               |

**Note 1:** Executing G51, G50, G28 and the manual machine zero return operation (machine zero return can automatically set the coordinate system) can automatically cancel offset value.

### 3.2.10 Feed per minute (G98)

**Command format:** G98 Fxxxx

**Function:** G98 is the feed per minute. The distance the tool traverses per minute is commanded by the numerical value following F, unit: mm/min.

**Explanation:** G98 is modal. Once G98 state is commanded, it is valid till G99 is executed.

### 3.2.11 Feed per Rev (G99)

**Command format:** G99 Fxxxx

**Function:** G99 is the feed per rev. The tool feed amount per the spindle rev is commanded by the numerical value following F, unit: mm/r.

**Explanation:** G99 is modal. Once G99 state is commanded, it is valid till G98 is executed.

Table 3-2-11-1 Difference between feed per minute and feed per rev

|                     | Feed per minute                                                                                                                                                                    | Feed per rev                           |
|---------------------|------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|----------------------------------------|
| Address designation | F                                                                                                                                                                                  | F                                      |
| Command designation | G98                                                                                                                                                                                | G99                                    |
| Range               | 1 mm/min~8000mm/min<br>(F1~F8000)                                                                                                                                                  | 0.001 mm/r~500.00mm/r<br>(F0.001~F500) |
| Limited value       | Feed per minute, feed per rev are limited in some speed. The limited values are set the machine manufacturer.<br>(The limited value is the numerical value following the override) |                                        |
| Override            | Feed per minute, feed per rev are adjusted by the override 0~150%.                                                                                                                 |                                        |

**Note 1:** When the position encoder's speed is below 1 r/min, the speed becomes uneven. The speed below 1 r/min can be used when the even speed is not required. Below 1r/min, the slower the speed is, the more uneven the speed is.

**Note 2:** G98, G99 are modal. They are valid after they are commanded till another command is executed.

**Note 3:** F code can permit inputting up to 7 digits. The system alarms when the input feedrate exceeds the limit.

**Note 4:** The position encoder must be installed on the spindle when the feed per rev is executed.

### 3.2.12 Constant Surface Control (G96, G97)

**Command format:** G96 S\_\_ ; constant surface control, and the numerical value following S specifies the surface speed of tool nose in the tangential direction.

G97 S\_\_ ; cancel the constant surface control code. The numerical value following S specifies the spindle speed.

**Function:** The constant surface is defined that the surface speed following S is constant. along the tool's position changing, the spindle speed is counted according to the previous given surface speed, the spindle speed is converted into the corresponding voltage to output the spindle control unit, which gets the constant relations between the instantaneous position of tool and the workpiece's surface.

**Explanation:** The surface speed unit is shown below:

Table 3-2-12-1

| Input unit | Surface speed unit |
|------------|--------------------|
| Metric     | m/min              |

The surface speed units are different according to the machine manufacturers.

When the constant surface control is executed, the rotary axis must be set on Z (X=0) of the workpiece coordinate system.

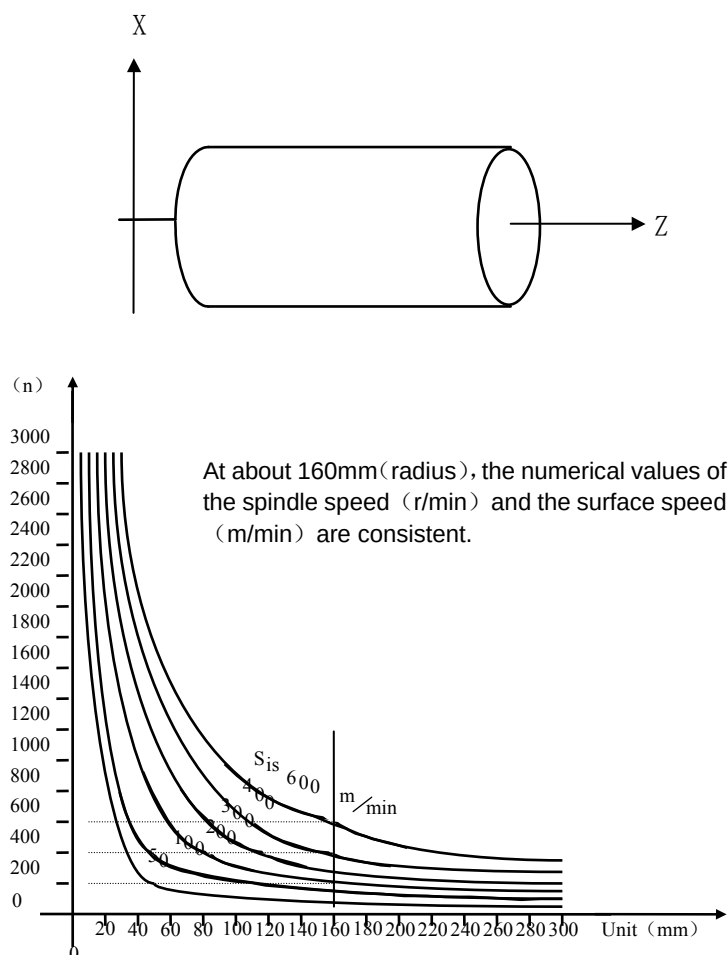


Fig. 3-2-12-1

#### 1) Spindle speed override

The specified surface speed or speed can use the override 50%, 60%, 70%, 80%, 90%, 100%, 110%, 120% according the spindle override selection.

#### 2) Max. spindle speed limit

Using the numerical value following G50 S can command max. spindle speed (r/min) in the constant surface control state.

G50 S\_\_ ;

In the constant surface control state, the spindle speed is limited to its max. speed when it exceeds the above specified value.

### 3) Constant surface control in G00

For the block in G00, when the constant surface speed control is executed, the system counts the surface speed at the end point of the block instead of the surface speed of the tool position which is changed constantly, which reason is that the rapid traverse does not execute the cutting.

**Note 1:** When the system is turned on, the state which does not set max. spindle speed is the unlimited state. For the limit, the system is only in G96 instead of G97.

**Note 2:** The lower limit value of the spindle speed in the constant surface speed control is controlled by P78. P78=100; when G50 S0 is executed, the spindle speed is 100 r/min.

**Note 3:** The commanded S value in G97 still retains in G96, i.e. it remain unchanged in G97. its value recovers when the system returns to G96 state.

G96 S50; (50m/min)

G97 S1000; (1000r/min)

G96 X3000; (50m/min)

**Note 4:** It does not run when the machine is locked, X coordinate value in the corresponding program changes and the system executes the constant surface control.

**Note 5:** The constant surface control is valid when thread cutting is executed, so, the constant surface control is invalid in G97, which makes the spindle rotate at the same speed.

**Note 6:** Feed per rev (G99) is still valid without useful meaning in G96.

**Note 7:** When G96 state is changed into G97 and there is no S code (r/min) in G97, the final speed in G96 is used as S code in G97.

N100 G97 S800; (800r/min)

...

N200 G96 S100; (100m/min)

...

N300 G97; (Xr/min)

X is speed of the block which is front of N300, i.e. when G96 state is changed into G97, the spindle speed does not change. When G97 state is changed into G96, S value in G96 is value. S=0 m/min when S value is not commanded.

## 3.2.13 Skip Function (G31)

**Command format:** G31 X (U) \_\_ Z (W) \_\_ F\_\_

**Function 1:** About the standard G31 function, the system skips to execute the next block when the external skip signal is input.

Example 1: G01 X0 Z0 F200

G31 X100 Z100 F100 when the block is running, the skip signal input is value, the system skips to the next block from the current block.

G01 X-50 Z-50 F150

**Function 2:** About the extended G31 function, the system skips to execute the next block when the external skip signal is input. The system continuously runs limitlessly in some direction when there is no skip signals.

Example 2: G01 X0 Z0 F200

G31 X100 Z100 F100 the system skips to execute the next block when the external skip signal is input validly. The system continuously runs limitlessly in some direction when there is no skip signals.

G01 X-50 Z-50 F150

**Example :**

Function 1: P145=0: when there is the skip signal input, the system skips to the next block from the current; the system runs to the position (X100, Z100) specified by the block, and then executes the next block when there is no skip signal input.

Function 2: P145=1: when there is the skip signal input, the system skips to the next block from the current; The system continuously runs limitlessly in some direction till the skip signal is input or the limit is executed when there is no skip signals.

**Note 1:** When G31 extension function is used, the system runs limitlessly in some direction, so, the hard limit should be installed on the machine.

**Note 2:** The input interface of skip signal is defined to XS39 12 when delivery.

### 3.2.14 Thread Cutting with Invariable Pitch (G32)

**Command format:** G32 X (U) \_\_ Z (W) \_\_ F (I) \_\_;

**Function:** Two axes moves simultaneously from the starting point (the position before G32 runs) to the end point specified X (U), Z (W) to execute the thread cutting, which is path shown in Fig. 3-2-14-1). The straight thread with invariable pitch, taper thread and end thread can be machine in G32. the tool retraction groove is needed in G32.

**Explanation:** X (U): absolute (incremental) coordinate of X thread cutting's end point;  
 Z (W): absolute (incremental) coordinate of Z thread cutting's end point;  
 F: metric thread pitch, i.e., the movement corresponding to the workpiece when the spindle rotates one rotation, range: 0.001 mm~500mm, it is modal;  
 I : tooth number per inch for inch thread, range: 0.06 tooth/inch~25400 tooth/inch, it is modal.

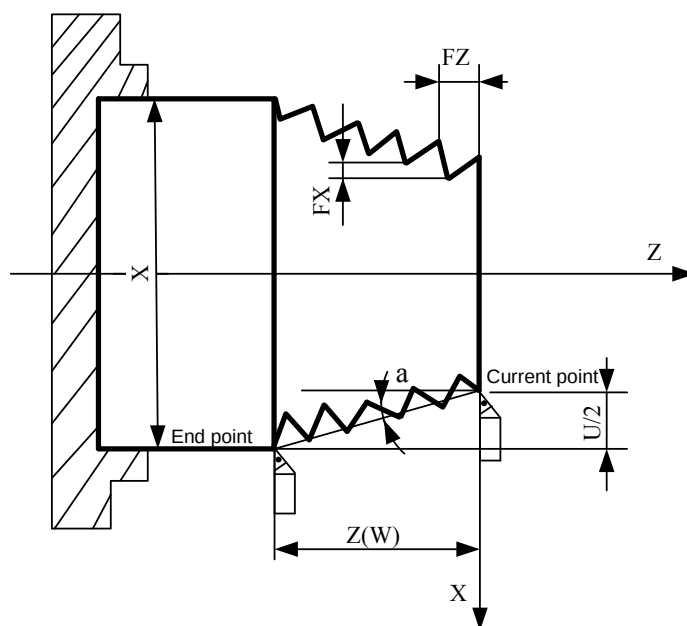


Fig. 3-2-14-1

The system uses a long axis and a short axis, which is shown in Fig.3-2-14-2.

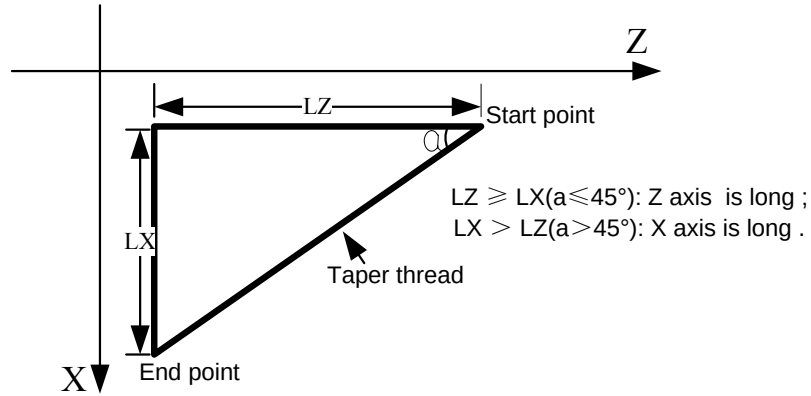


Fig. 3-2-14-2

Notes:

1. Start and end of thread cutting, some lead is wrong because of acceleration and deceleration, the commanded thread length should be longer than the required, which is shown in Fig. 3-2-14-3.
2. The federate override is invalid, and is the constant value 100% in the course of thread cutting.
3. The spindle must be started when thread cutting, otherwise, the system alarms; the spindle cannot stop in the course of thread cutting..
4. The spindle override is invalid in the course of the thread cutting. Changing the spindle override brings wrong thread because of acceleration/deceleration.
5. The feed hold function is invalid in the course of the thread cutting. After he first non thread cutting block is executed when the thread cutting is executed, stopping the single block run stops the thread cutting.
6. When the previous block is for thread cutting and the current is also for it, the system does not check the one rotation signal of the spindle position encoder at beginning of thread cutting.
7. The spindle speed must be constant. When the spindle speed changes, the thread appears deviation.
8. The system alarms when F, I are in the same block simultaneously.

Example 1: Using G32 compiles a program which path is shown in Fig. 3-2-14-3, its thread pitch: 4mm

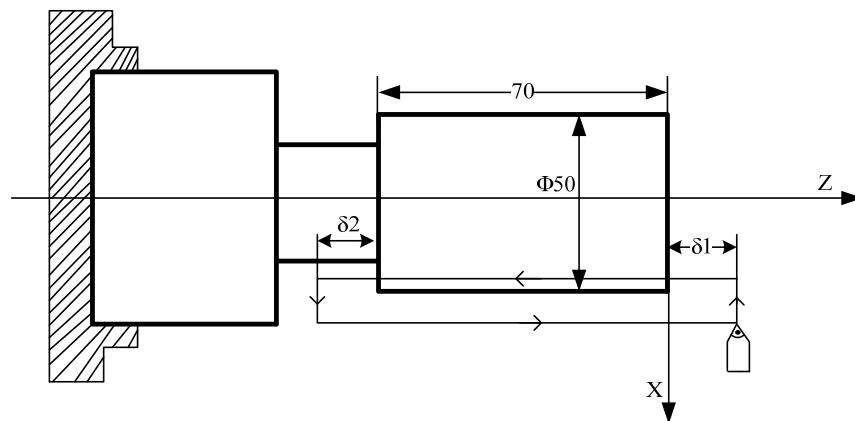


Fig. 3-2-14-3

A program:  $\delta 1 = 3\text{mm}$ ,  $\delta 2 = 1.5\text{mm}$ , tool cutting depth 1mm (single side) , cut-in two times.

G0 X100 Z50; (rapid positioning)

M03 S200; (start spindle, speed 200)

T0101; (call thread tool)

```

G0 X49 Z3;    (rapid positioning, the first cut-in 1mm)
G32 W-74.5 F4.0;
G00 X55;
W74.5;
X48;          (rapid positioning, the second cut-in 1mm)
G32 W-74.5 F4.0;
G00 X55
W74.5;
G0 X100 Z50 M05;
M30;

```

Example 2: Using G32 compiles a program which path is shown in Fig. 3-2-14-4, the length axis is Z, it thread pitch: 3mm

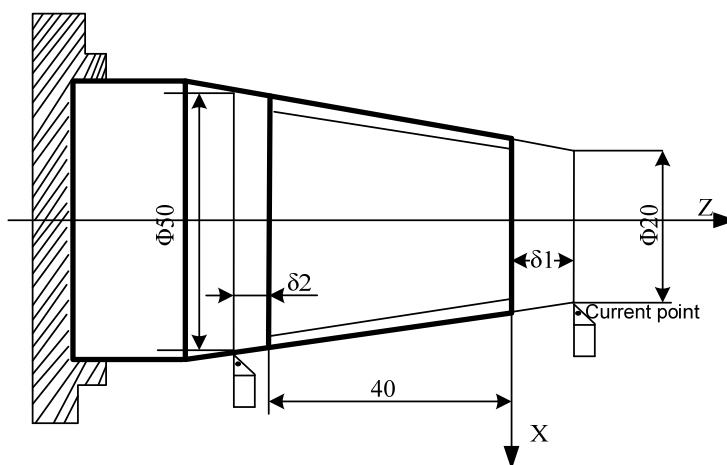


Fig. 3-2-14-4

A program:  $\delta 1 = 2\text{mm}$ ,  $\delta 2 = 1\text{mm}$ , tool cutting depth 1mm (single side), cut-in two times.

```

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

```

### 3.2.15 Thread Cutting with Variable Pitch (G34)

**Command format:** G34 X (U) \_\_ Z (W) \_\_ F (I) \_\_ K \_\_;

**Function:** Two axes moves simultaneously from the starting point (the position before G34 runs) to the end point specified X (U), Z (W) to execute the thread cutting. The straight thread with variable pitch, taper thread and end thread can be machine in G32. the tool retraction groove is needed in G34.

**Explanation:** X (U): absolute (incremental) coordinate of X thread cutting's end point;

Z (W): absolute (incremental) coordinate of Z thread cutting's end point;

F: metric thread pitch, pitch of thread's starting point, range: 0.001 mm~500mm, it is modal;

I: tooth number per inch for inch thread, range: 0.06 tooth/inch~25400 tooth/inch, it is modal;

K: pitch's increment or decrement of one rotation of the spindle. K range:  $\pm 0.000001\text{mm} \sim \pm 500\text{mm}$ .

$\pm 0.000001\text{ inch} \sim \pm 19.685\text{ inch}$

When K value increasing or decreasing makes pitch exceed the permissive value or reduce to 0 or the negative value, the system alarms; at the same time, when the pitch changes greatly, the acceleration/deceleration in the course of thread machining greatly becomes slow, which cause the wrong pitch.

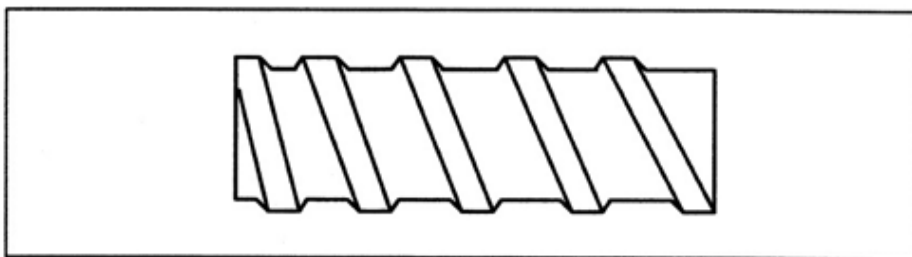


Fig. 3-2-15-1

**Note:** In MDI mode, I, K value can be input and cannot be displayed. Do not run G34 in MDI mode.

### 3.2.16 Tapping Cycle (G33)

**Command format:** G33 Z (W) \_\_ F (I) \_\_; G33X(U)\_\_F(I)\_\_;

**Function1:** Z moves from starting point (the position before G33 runs) to end point specified by Z (W) to execute the rigid tapping (its path is shown in Fig.3-2-16-1), the spindle rotates reversely and Z returns to the starting point of the cycle.

**Function2:** X moves from starting point (the position before G33 runs) to end point specified by X (U) to execute the rigid tapping, the spindle rotates reversely and X returns to the starting point of the cycle. (Its execution is the same that of Z but its direction is different from that of X)

**Explanation:** Z (W): absolute (incremental) coordinate of Z thread cutting's end point;

X (U): absolute (incremental) coordinate of X thread cutting's end point;

F: metric thread pitch, i.e. the tool movement corresponding to the workpiece when the spindle rotates one rotation, range: 0.001 mm~500mm, it is modal;

I: tooth number/inch for inch thread, range: 0.06 tooth/inch~25400 tooth/inch, it is modal.

**G33 Z tapping cycle process:**

- 1) Z tool infeed tapping;
- 2) Stop the spindle;
- 3) Wait the spindle to exactly stop;
- 4) The spindle rotates reversely; (opposite to previous rotation direction)
- 5) Z executes the tool retraction to starting point of machining;
- 6) The spindle remains rotation, but the rotation direction and the tool retraction direction are consistent, G33 runs completely;

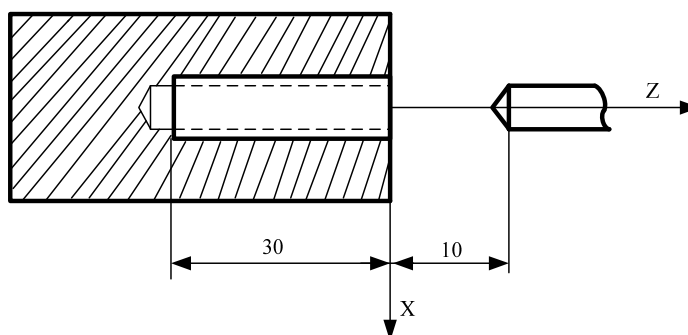


Fig. 3-2-16-1 G33 tapping cycle

**Example:** single-thread with pitch 2mm

```

N0010 M3 S300;      start the spindle
N0020 G00 X0 Z3;    rapid positioning
N0030 G33 Z-30 F2 ; tapping cycle
N0040 G0 Z100 X30 ; return to starting point
N0050 M30 ;        end of program

```

**Note 1:** G33 is rigid tapping. The spindle decelerates in some time after the spindle stop signal is valid. At the moment, Z rotates along with the spindle till the spindle exactly stops, the bottom hold of the thread when the actual machining should be deeper than the actual demand, and the concrete length is determined by the spindle speed when tapping and by whether there is spindle brake.

**Note 2:** Other notes are the same those of G32.

**Note 3:** G33 are modal.

**Note 4:** I value can be input in MDI mode and cannot be displayed. Do not execute G33 in MDI mode.

### 3.3 Single Fixed Cycle Command

For some special roughing, the same machining path are cut repetitively because of much cutting, the system can use a fixed cycle function, adopting a block to complete the machining operations by many blocks. When the repetitive cutting is executed, the corresponding numerical value is changed, which is valid to simply programs. The single fixed cycle commands include outer (inner) cutting cycle G90, thread cutting cycle G92 and end cutting cycle G94.

The diameter is specified in the following figures. When the radius is specified, U/2 replaces U, X/2 replaces X.

### 3.3.1 Outer (Inner) Cutting Cycle (G90)

**Command format:** G90 X (U) \_\_ Z (W) \_\_ R\_\_ F\_\_;

**Function:** Using G90 can realize the single cycle machining of cylindrical surface, and taper surface, and the tool returns to the starting point after the cycle is completed. The dotted lines(R) (R) in Fig.3-3-1-1, Fig. 3-3-1-2 mean to be the rapid traverse, and the real lines (F) means to be cutting feed. The sign of numerical value following the address U is determined by X direction of the path 1 and the one following W is determined by Z direction of the path 2.

**Explanation:** X,Z: absolute coordinates of circular end point, unit: mm;

U,W: coordinates of circular end point corresponding to the circular starting point, unit: mm;

R: it is radius difference between starting point and end point of taper cutting, unit: mm;

F: it is combined speed of X, Z in the cycle, and it is modal.

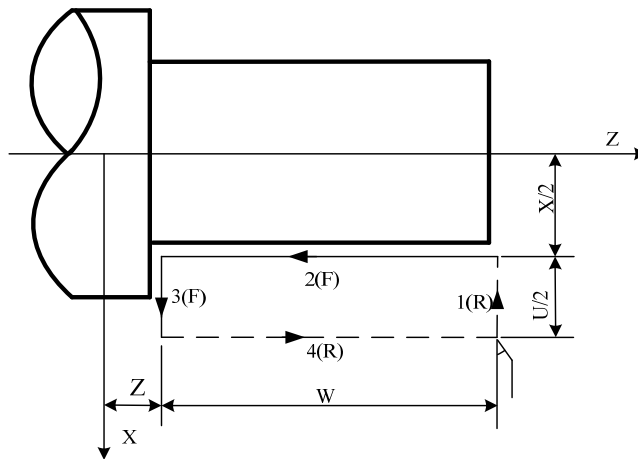


Fig. 3-3-1-1

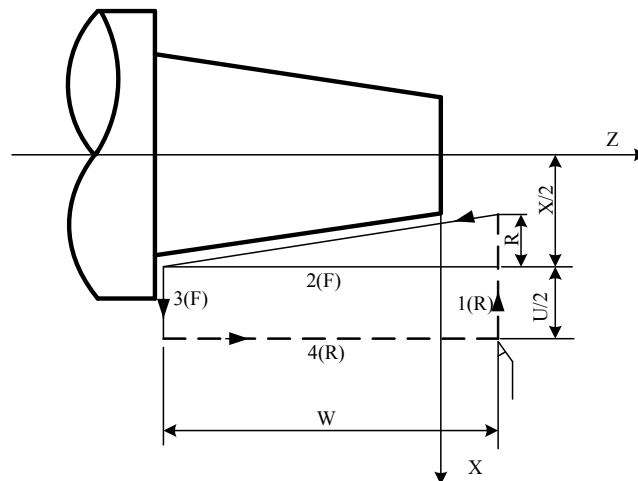


Fig. 3-3-1-2

G90 has four kind of path, which are shown in Fig. 3-3-1-3 according to the different start-up tool positions.

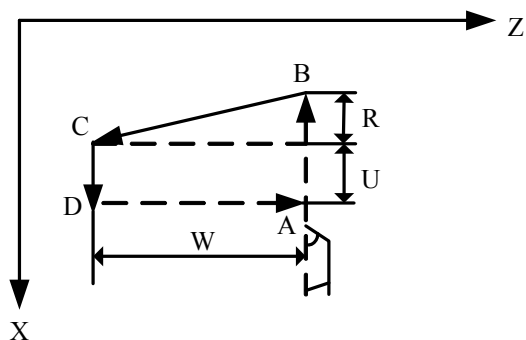
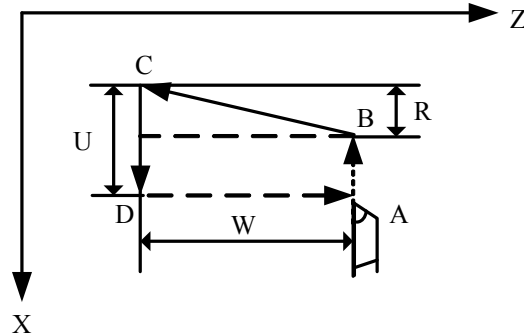
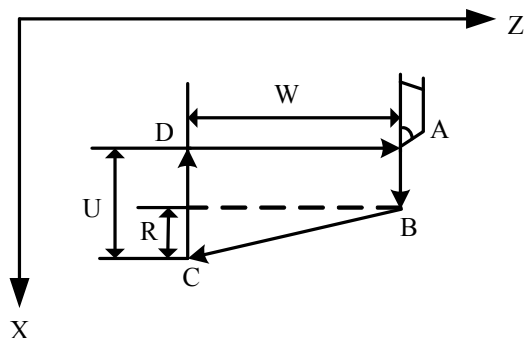
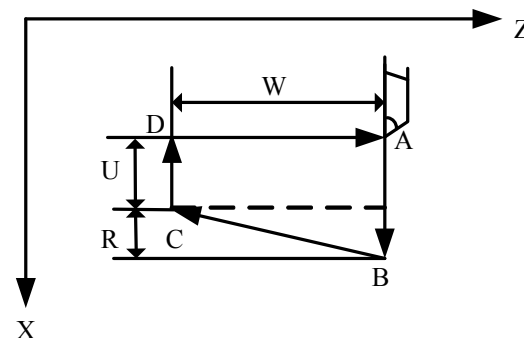
1)  $U < 0, W < 0, R < 0$ 

2)  $U < 0, W < 0, R > 0$  but  $|R| = |U/2|$ 

3)  $U > 0, W < 0, R < 0$  but  $|R| = |U/2|$ 

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


Fig.3-3-1-3 G90 path

Example: using 90 compiles a program which path is shown in Fig. 3-3-1-4.

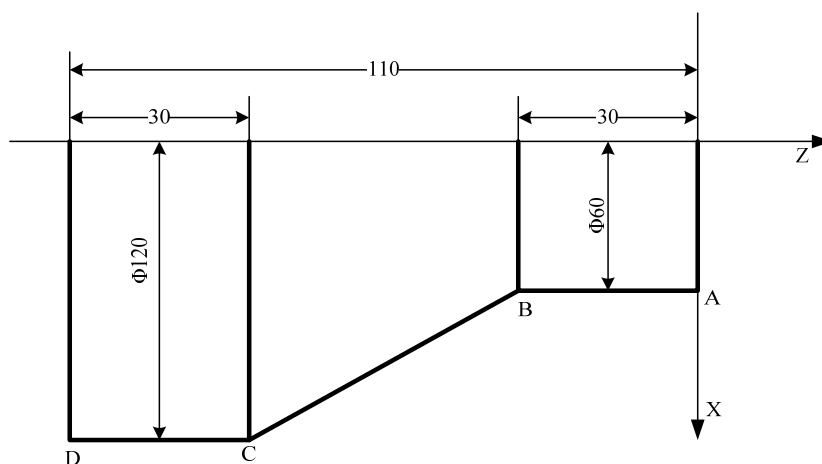


Fig. 3-3-1-4

A program:

```
O0001;
M3 S300;
G0 X130 Z5;
G90 X120 Z-110 F200; (C→D)
```

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

### 3.3.2 End Cutting Cycle (G94)

**Command format:** G94 X (U) \_\_ Z (W) \_\_ R\_\_ F\_\_;

**Function:** Executing G94 can perform a single cycle machining of end, the tool returns to the starting point after the cycle is completed.

R means the rapid traverse in Fig.3-3-2-1. Fig. 3-3-2-2, F means the cutting feed. The sign of numerical value following U in incremental programming is determined by X direction of path 2, that following W is done by Z direction of path 1.

**Explanation:** X, Z: absolute coordinate values of circular end point, unit: mm;

U, W: coordinates of circular end point corresponding to circular starting point, unit: mm;

R: Z coordinate component from starting point of end cutting to end point displacement, unit: mm;

F: it is combined federate of X, Z, it is modal.

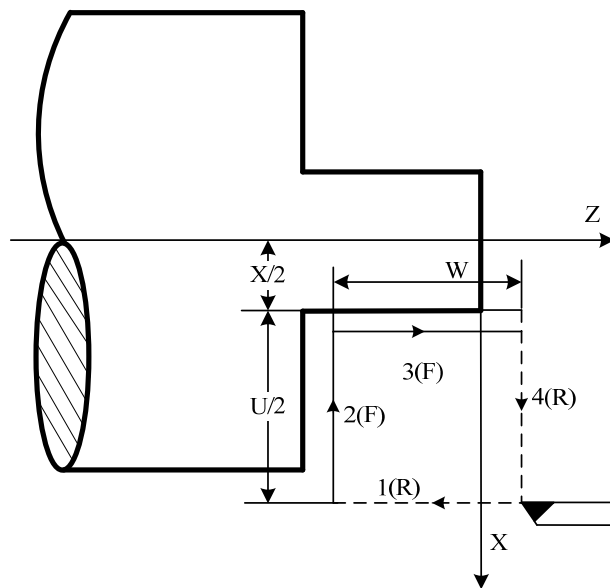


Fig. 3-3-2-1

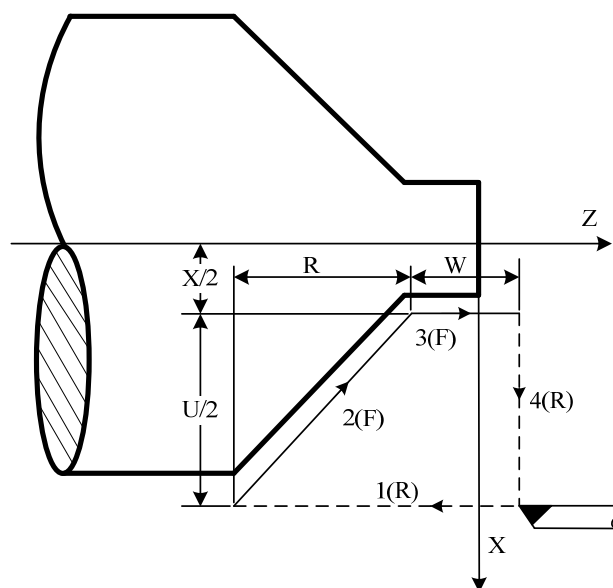


Fig. 3-3-2-2

G94 has four kind of path according to start-up tool position, which is shown in Fig. 3-3-2-3:

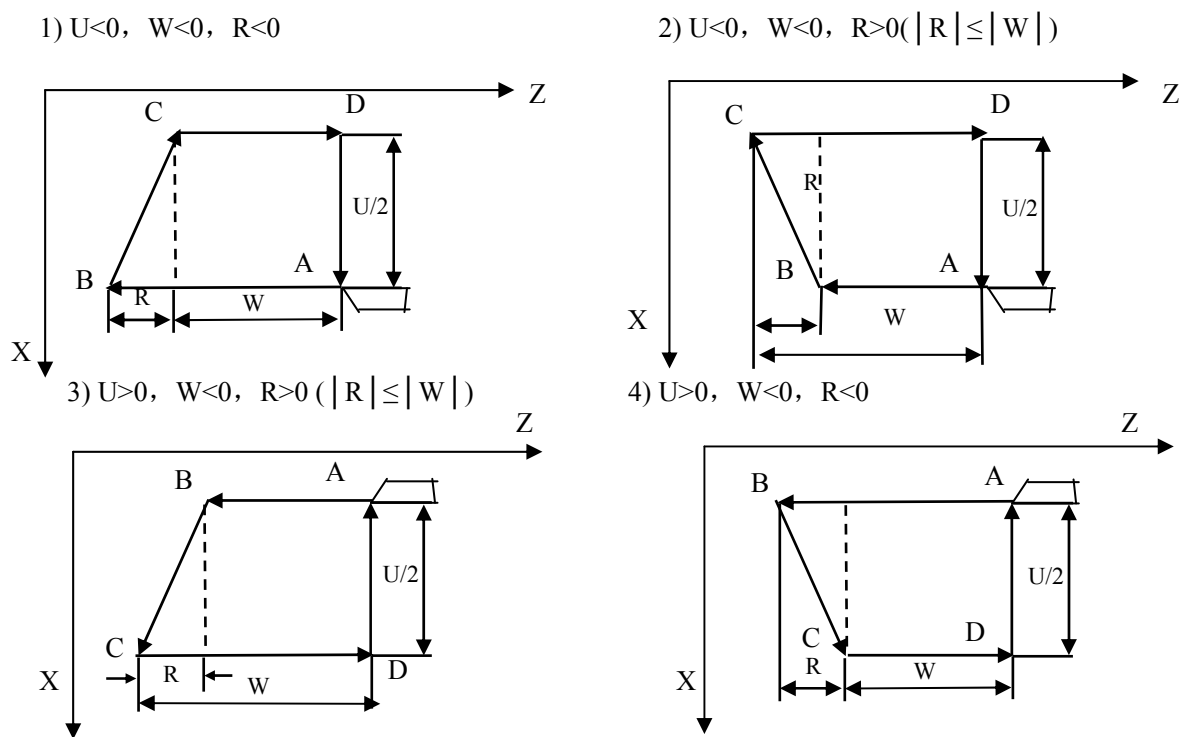


Fig. 3-3-2-3 G94 path

Example: using G94 compiles a program shown in Fig. 3-3-1-4.

A program:

```
O0002;
M3 S1;
G0 X130 Z5;
G94 X120 Z-110 F100; (D→C)
G0 X120 Z0;
```

```
G94 X60 Z-30 R-50; (C→B→A)
M5 S0;
M30;
```

### 3.3.3 Thread Cutting Cycle (G92)

#### Command format:

G92X(U)\_\_\_Z(W)\_\_\_J\_\_\_K\_\_\_F\_\_\_L\_\_\_P\_\_\_; (metric thread)

↑  
Assign thread pitch(F)

G92X(U)\_\_\_Z(W)\_\_\_J\_\_\_K\_\_\_I\_\_\_L\_\_\_P\_\_\_; (inch thread)

↑  
Assign thread pitch (tooth/inch)

**Function:** Executing G92 can machine the straight thread with constant pitch and the single cycle thread machining of taper thread, the tool returns to the starting point after the cycle is completed. The grooving is not demanded when thread cutting. When the user does not use J, K to set length of thread run-out, the thread run-out length=set value set by P68 X0.1X pitch. In incremental programming, the sign of numerical value following U is determined by X direction of path 1, that following W is done by Z direction of path 2. Dotted lines in Fig. 3-3-3-1 and Fig. 3-3-3-2 means rapid traverse, real lines means cutting feed. When J, K sets values, G92 executes X, Z thread run-out according to J, K set value; when only J or K value is set, 45° thread run-out is executed.

**Explanation:** X,Z: coordinats of circular end point, unit: mm;

U,W: coordinates of circular end point corresponding to circular starting point, unit: mm;

J: proportion of X thread run-out, without sign. Unit: mm, J is the radius designation;

K: proportion of Z thread run-out, without sign. Unit: mm;

R: radius difference between thread starting point and end point, unit: mm;

F: metric thread pitch, range: 0.001~500, unit: mm, modal;

I: inch thread/inch tooth, range: 0.06~25400, unit: tooth/inch, modal;

L: number of thread head, range: (1~99), unit: head, modal; it is 1 when it is not assigned;

P: length of thread run-out, range: (0~255), unit: 0.1 pitch;

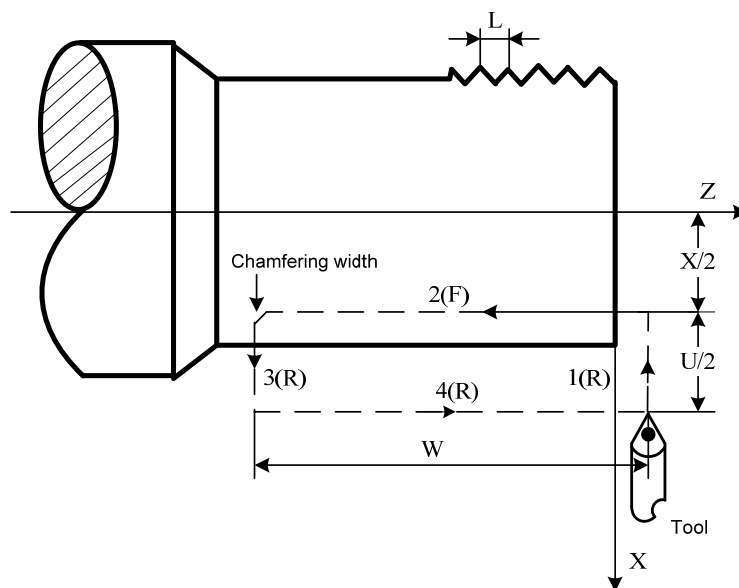


Fig. 3-3-3-1

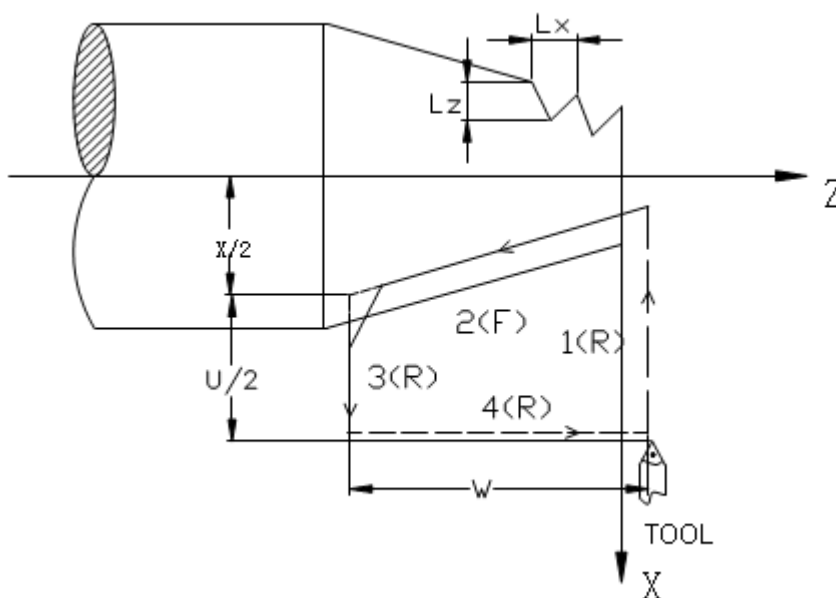


Fig. 3-3-3-2

**Notes:**

1. Notes of G92 are the same those of G32, refer to Section 3.2.14.
2. When there is the feed hold signal (pause) input in thread cutting cycle in the thread cutting cycle, the cycle is executed till the operations of 3 completes to stop.
3. Thread pitch range, spindle speed limit are the same those of G32.
4. When using G92 machines the straight thread, start-up tool point and thread end point are the same in X direction, the system alarms because the system cannot identify the inner thread or outer.
5. R value in G92 is referred to Fig. 3-3-1-3.
6. When one of J, K is set to 0, or is not assigned, the system takes it as 45° thread run-out.
7. I value can be input in MDI mode and cannot be displayed. Do not execute G92 in MDI mode.

**Example:** using G90 compiles a program shown in Fig. 3-3-3-3, and using G92 executes machining thread.

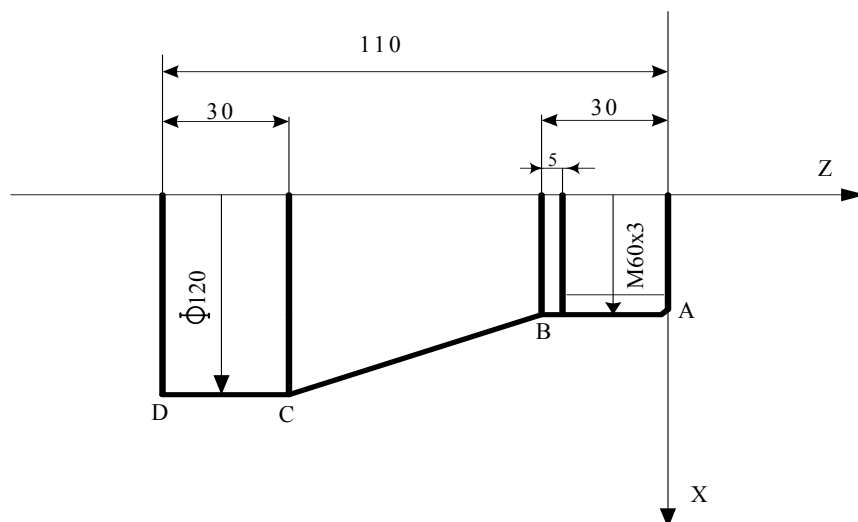


Fig. 3-3-3-3

A program:

```

O0001;
M3 S300;
G0 X150 Z50;
T0101;                (outer turning)
G0 X130 Z5;
G90 X120 Z-110 F200;   (C→D)
X60 Z-30;             (A→B)
G0 X130 Z-30;
G90 X120 Z-80 R-30 F150; (B→C)
G0 X150 Z150;
T0202;                (thread cutter)
G0 X65 Z5;
G92 X58.5 Z-25 F3;     (machine thread, 4 times cut-in)
X57.5 Z-25;
X56.5 Z-25;
X56 Z-25;
M5 S0;
M30;

```

### 3.3.4 Notes in Single Fixed Cycle Commands

- 1) In the single fixed cycle, X(U), Z(W), R are modal. When the new X(U), Z(W), R are assigned, the previously commanded data are valid.

- 2) In the single fixed cycle, X (U), Z (W), R are cleared when the non modal G codes except G04 or other commands except for G90, G92, or G94 in Group 01 are commanded.
- 3) When there is only blocks without movement codes after G90, G92, G94, the fixed cycle is not executed repetitively.

(Example) N003 M3;

...

...

N010 G90 X20.0 Z10.0 F2000;

N011 M8; (do not repetitively execute G90)

...

...

- 4) In the fixed cycle, when M, S, T are commanded, the fixed cycle and M, S, T are executed simultaneously. Cancel the fixed cycle after M, S, T is executed, execute it again.

(Example) N003 T0101;

...

N010 G90 X20.0 Z10.0 F2000;

N011 G00 T0202;

N012 G90 X20.5 Z10.0;

### 3.4 Compound Fixed Cycle Commands

To simply programming, the system provides six compound cycle commands, including outer (inner) roughing cycle G71; end roughing G72, closed cutting cycle G73/ finishing machining G70; end deep hole machining cycle G74; outer grooving cycle G75 and compound thread cutting cycle G76. The system only specifies tool cutting values of finishing and roughing, and automatically counts machining paths and number of cutting.

#### 3.4.1 Outer (Inner) Roughing Cycle (G71)

**Command format:** G71u ( $\Delta d$ ) R (e);

G71 P (NS) Q (NF) U ( $\Delta u$ ) W ( $\Delta w$ ) F S T ;

N (NS) . . . . . ;

. . . . . ;

. . . . F;

. . . . S;

. . . . T;

.

N (NF) . . . . . ;

} blocks for finishing path

**Function:** According to the workpiece's finishing path, cutting amount, tool infeed and tool retraction specified by the block, the system automatically count the roughing path, which is shown in Fig.3-4-1-1. Cutting is executed by operations which are parallel to Z. The non forming rod can be formed one time.

**Explanation:**  $\Delta d$ : cutting depth without sign at one time. The cut-in direction is determined by AA' (specified by the radius), range: 0.001mm~9999.999mm. It is modal and valid before the next is specified. It can be specified by P71, the parameter value also changes according to the program command.

e: Tool retraction (specified by the radius), unit: mm, range: 0.001mm~9999.999mm. It is modal and valid before the next is specified. It can be specified by P72, the parameter value also changes according to the program command.

NS: It is the first block's serial number in blocks for finishing path.

NF: It is the last block's serial number in blocks for finishing path.

$\Delta u$ : Distance and direction of X finishing allowance, range: -9999.999mm~9999.999mm.

$\Delta w$ : Distance and direction of Z finishing allowance, range: -9999.999mm~9999.999mm.

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

S: Spindle speed;

T: Tool, tool offset number;

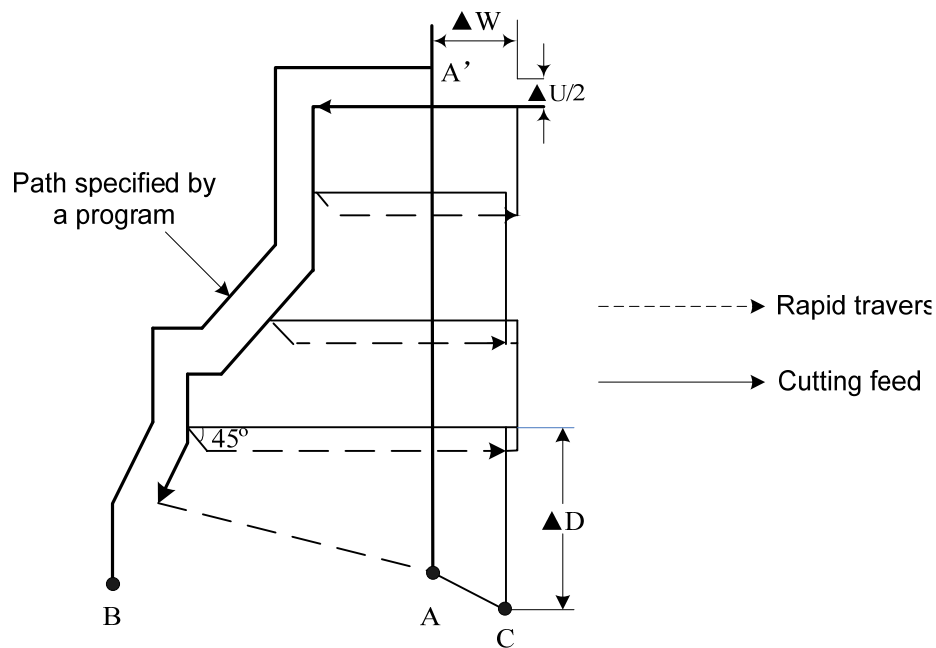


Fig. 3-4-1-1 G71 path

1.  $\Delta d$ ,  $\Delta u$  are specified by the same address U, and their difference is whether their blocks specify P, Q.
2. The circular operations are executed G71 specified by P, Q.
3. In G71, F, S, T function in the blocks NS~NF are invalid and all are neglected. G71's blocks or the previous specified F, S, T are valid. F, S, T in the NS~NF are valid only to G70.
4. With the constant surface speed control selection function, G96 or G97 in the blocks NS~NF is invalid, G71 or the previous blocks are valid.

5. According to the cut-in directions, G71 has four paths (Fig.3-4-1-2), which are executed according to operations which are parallel, signs of  $\Delta u, \Delta w$  are shown below:

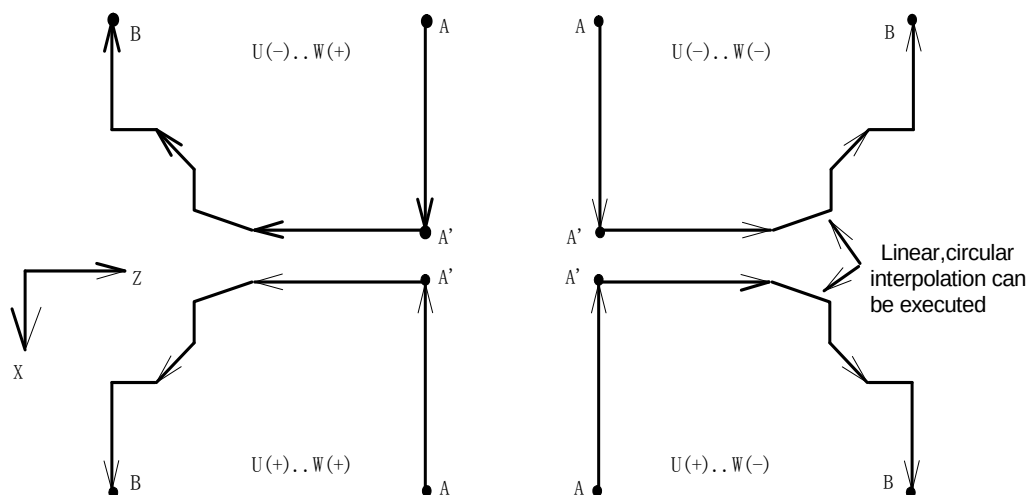


Fig. 3-4-1-2

6. Only G00 or G01 is specified in the blocks which serial number is NS between A and A', and coordinates of A and A' are consistent.
7. Between A' and B, X,Z values must monotonously increase or decrease, but their path cannot be monotonous.
8. The system cannot call subprograms between NS and NF.
9. There are most 100 blocks between NS and NF, ERR137 alarm occurs when the blocks exceed 100.

Example: Using G71 compiles a program shown in Fig.3-4-1-3.

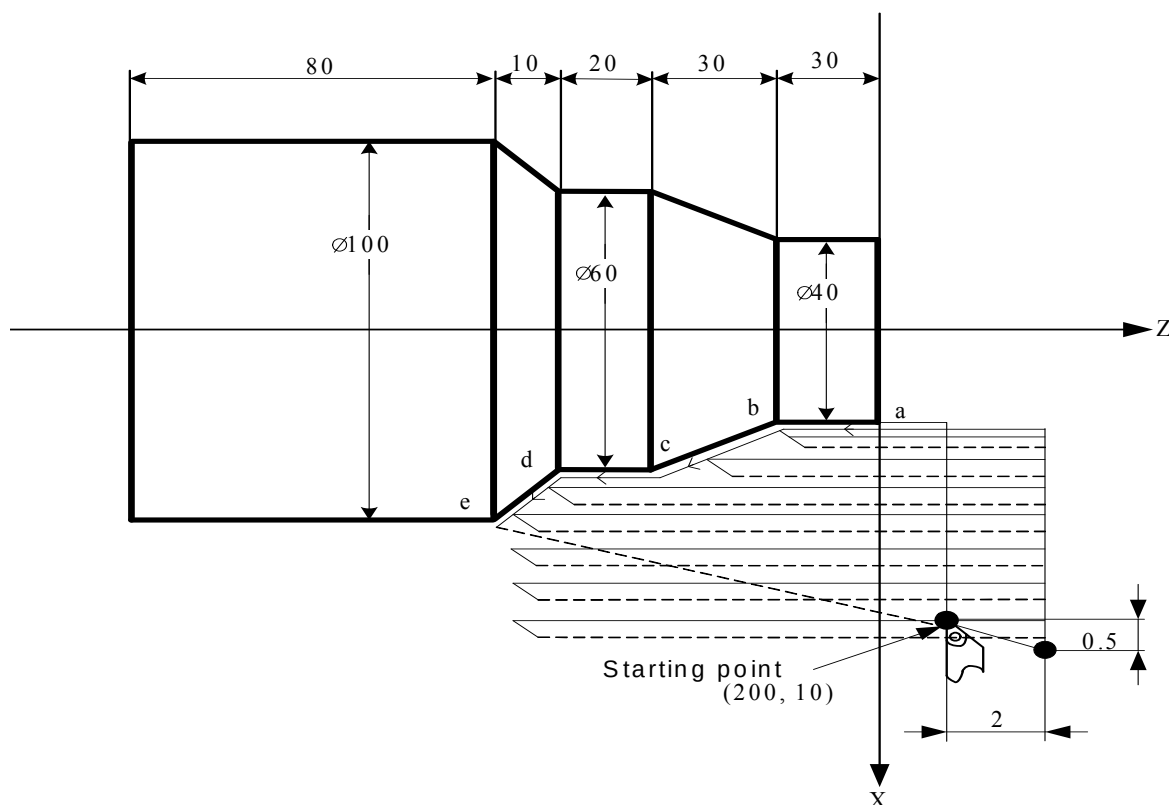


Fig. 3-4-1-3

A program:

|                                       |                                                                                                         |
|---------------------------------------|---------------------------------------------------------------------------------------------------------|
| O0001;                                |                                                                                                         |
| N010 G0 X220.0 Z50;                   | (positioning to safety position)                                                                        |
| N020 M3 S300;                         | (the spindle rotating CCW, speed 300r/min)                                                              |
| N030 M8;                              | (cooling ON)                                                                                            |
| N040 T0101;                           | (use a roughing tool )                                                                                  |
| N050 G00 X200.0 Z10.0;                | (rapid positioning, approach the workpiece)                                                             |
| N060 G71 U0.5 R0.5;                   | (cut-in 1mm (diameter) at one time, tool retraction<br>1mm[diameter])                                   |
| N070 G71 P080 Q120 U1 W2.0 F100 S200; | { (a→d roughing, X allowance 1mm, Z 2mm)<br>(positioning to X40)<br>blocks for finishing path a→b→c→d→e |
| N080 G00 X40.0;                       |                                                                                                         |
| N090 G01 Z-30.0 F100 S200; (a→b)      |                                                                                                         |
| N100 X60.0 W-30.0; (b→c)              |                                                                                                         |
| N110 W-20.0; (c→d)                    |                                                                                                         |
| N120 X100.0 W-10.0; (d→e)             |                                                                                                         |
| N130 G00 X220.0 Z50.0;                | (execute rapid tool retraction to safety position)                                                      |
| N140 T0202;                           | (use No. 2 finishing tool, execute No 2. tool offset)                                                   |
| N150 G00 X200.0 Z10.0;                | (positioning to the cycle starting point of G7)                                                         |
| N160 G70 P80 Q120;                    | (finishing a--- e)                                                                                      |
| N170 M05 S0;                          | (stop the spindle, speed)                                                                               |
| N180 M09;                             | (cooling OFF)                                                                                           |
| N190 G00 X220.0 Z50.0 T0100;          | (rapid return to safety position, use reference tool, clear tool offset)                                |
| N200 M30;                             | (end of program)                                                                                        |

### 3.4.2 End Roughing Cycle (G72)

**Command format:** G72 W (Δd) R (e);

G72 P (NS) Q (NF) U (Δu) W (Δw) F\_\_ S\_\_ T\_\_;

|                    |   |                           |
|--------------------|---|---------------------------|
| N (NS) . . . . . ; | } | blocks for finishing path |
| . . . . . ;        |   |                           |
| . . . . F;         |   |                           |
| . . . . S;         |   |                           |
| . . . . T;         |   |                           |
| . . . . .          |   |                           |
| N (NF) . . . . . ; |   |                           |

**Function:** According to the workpiece's finishing path, cutting amount, tool infeed and tool retraction specified by the block, the system automatically count the roughing path Cutting is executed by operations which are parallel to X. The non forming rod can be formed one time.

**Explanation:**

Δd: cutting depth without sign at one time. The cut-in direction is determined by AB (specified by the

- radius), range: 0.001mm~9999.999mm. It is modal and valid before the next is specified. It can be specified by P71, the parameter value also changes according to the program command
- e: Tool retraction, unit: mm, range: 0.001mm~9999.999mm. It is modal and valid before the next is specified. It can be specified by P72, the parameter value also changes according to the program command.
- NS: It is the first block's serial number in blocks for finishing path.
- NF: It is the last block's serial number in blocks for finishing path.
- $\Delta u$ : Distance and direction of X finishing allowance, range: -9999.999mm~9999.999mm.
- $\Delta w$ : Distance and direction of Z finishing allowance, range: -9999.999mm~9999.999mm.
- F: Cutting feedrate, range: feed/min : 1mm/min~8000mm/min, feed/r: 0.001mm/r~500mm/r.
- S: Spindle speed;
- T: Tool, tool offset number;

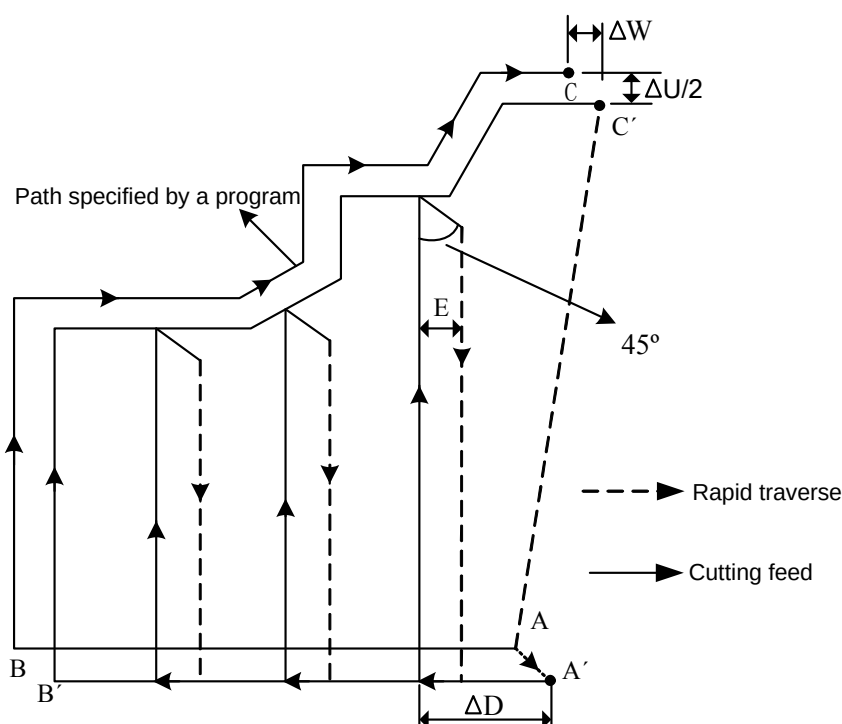


Fig. 3-4-2-1

**Explanations:**

1.  $\Delta d$ ,  $\Delta u$  are specified by the same address W, and their difference is whether their blocks specify by P, Q.
2. The circular operations are executed G72 specified by P, Q.
3. In G72, F, S, T function in the blocks NS~NF are invalid and all are neglected. G72's blocks or the previous specified F, S, T are valid. F, S, T in the NS~NF are valid only to G70.
4. With the constant surface speed control selection function, G96 or G97 in the blocks NS~NF is invalid, G72 or the previous blocks are valid.
5. According to the cut-in directions, G72 has four paths (Fig.3-4-2-2), which are executed according to operations which are parallel, signs of  $\Delta u$ ,  $\Delta w$  are shown Fig.3-4-2-2.
6. Only G00 or G01 is specified in the blocks which serial number is NS between A and B, and coordinates of A and B are consistent.
7. Between B and C, X,Z values must monotonously increase or decrease, but their path cannot be monotonous.

8. The system cannot call subprograms between NS and NF.
9. There are most 100 blocks between NS and NF, ERR137 alarm occurs when the blocks exceed 100.

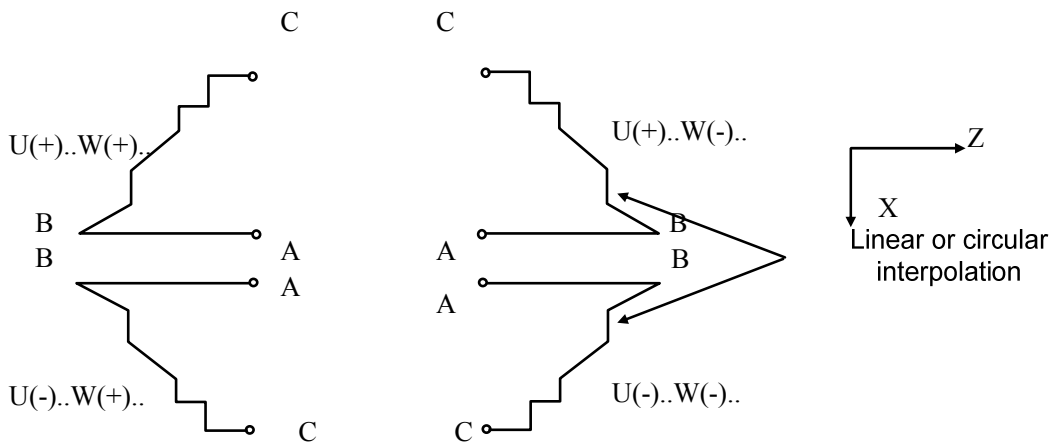


Fig. 3-4-2-2 four shapes of G72 path

Example: using compound fixed cycle G72 compiles a program shown in Fig. 3-4-2-3.

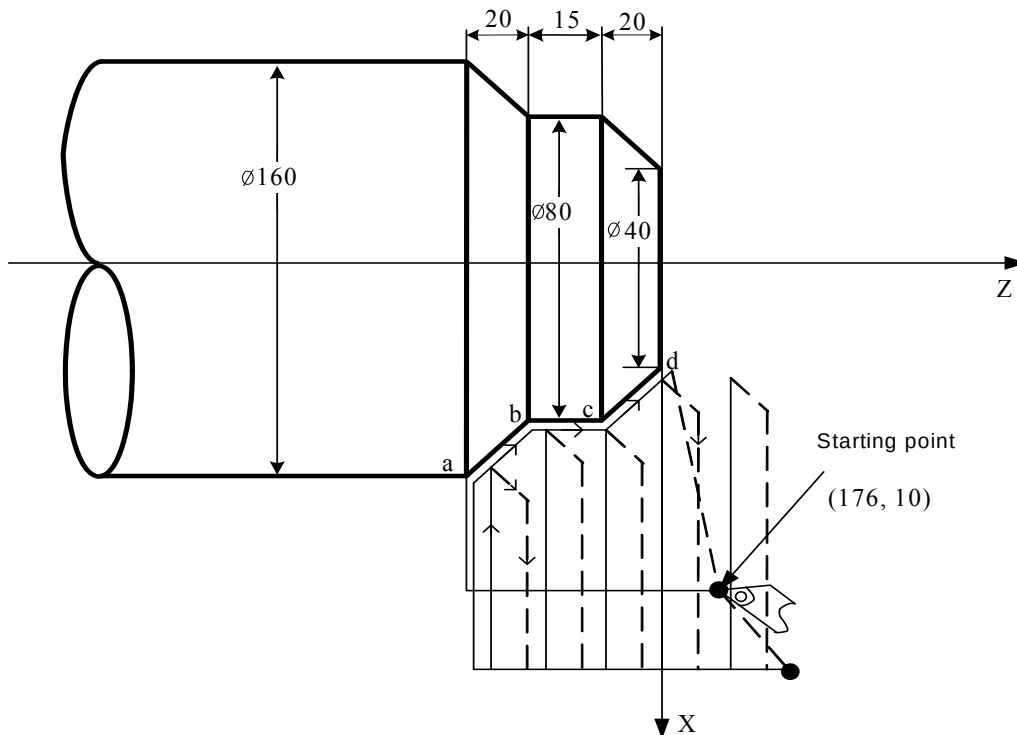


Fig. 3-4-2-3

Program:

```

O0002;
N010 G0 X220.0 Z50.0; (position to safety position)
N015 T0202;           (execute No. 2 tool, No. 2 tool offset)
N017 M03 S200;        (the spindle rotating CCW, speed 200)
N020 G00 X176.0 Z10.0; (rapid positioning, approach the workpiece)
N030 G72 W2.0 R1.0;   (tool infeed 2mm, tool retraction 1mm)
    
```

```

N040 G72 P050 Q090 U1.0 W1.0 F100 S200;    (a—d roughing, X allowance 1mm, Z allowance
1mm)
N050 G00 Z-55.0 S200 ; (rapid positioning)
N060 G01 X160.0 F120;    (tool infeed to point a)
N070 X80.0 W20.0;        (machining a—b)
N080 W15.0;              (machining b—c)
N090 X40.0 W20.0 ;        (machining c—d)
N100 G0 X220.0 Z50.0;    (rapid tool retraction to safety position)
N105 T0303;              (execute No. 3 tool, No. 3 tool offset)
N108 G00 X176.0 Z10.0;   (rapid return G70)
N110 G70 P050 Q090;      (finishing a—d)
N115 G0 X220.0 Z50.0;    (traverse to safety position to execute tool change)
N120 M5 S0 T0200;        (stop the spindle, execute No. 2 tool and cancel tool compensation)
N130 G0 X220.0 Z50.0;    (rapid return to starting point)
N140 M30;                (end of program)

```

} blocks for finishing path

### 3.4.3 Closed Cutting Cycle (G73)

**Command format:** G73u (Δi) W (Δk) R (d);

G73 P (NS) Q (NF) U (Δu) W (Δw) F S T ;

**Function:** G73 can execute the repetitive cutting along the path specified by the blocks NS~NF, the tool moves forward once every cutting. For the smithing, casting and other roughing has formed preliminarily, which can improve the machining efficiency.

**Explanation:** Δi: distance and direction of X tool retraction (radius value), unit: mm; it is modal and valid before the next is specified. It can be specified by P73, the parameter value also changes according to the program command.

Δk: Distance and direction of Z tool retraction (radius value), unit: mm; it is modal and valid before the next is specified. It can be specified by P74, the parameter value also changes according to the program command.

D: Times of closed cutting, unit: times; it is modal and valid before the next is specified. The parameter value also changes according to the program command, when set by P75.

NS: It is the first block's serial number in blocks for finishing path;

NF: It is the last block's serial number in blocks for finishing path;

Δu: Distance and direction of X finishing allowance, range: -9999.999mm~9999.999mm. the specified value is the radius or diameter value according to N0:1#3;

Δw: Z finishing allowance, range: -9999.999mm~9999.999mm;

F: Cutting feedrate, range: 1 mm/min~8000mm/min;

S: Spindle speed;

T: Tool, tool offset number;

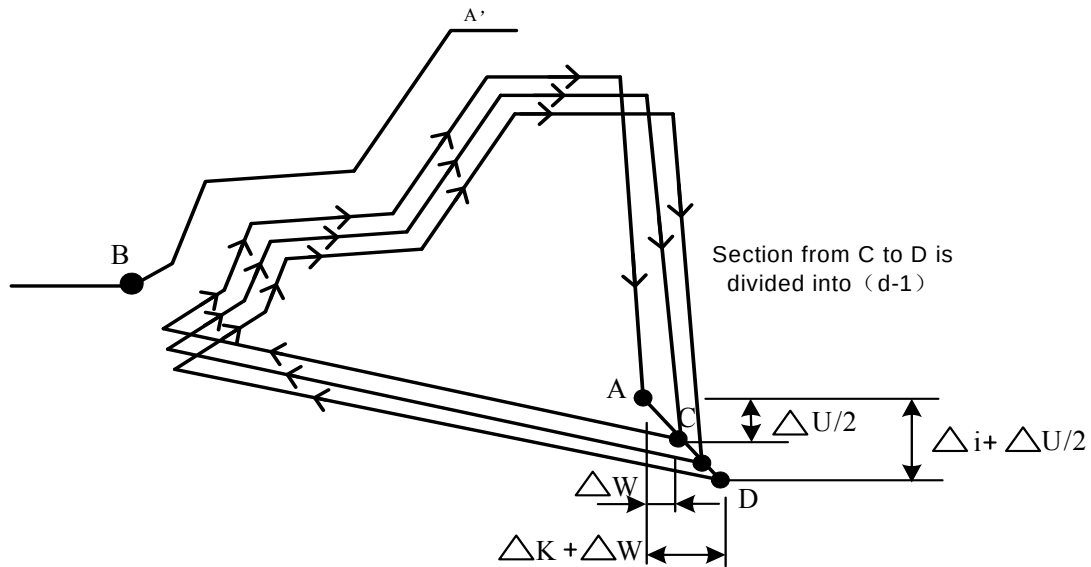


Fig. 3-4-3-1 G73 path

1. F, S, T functions in any blocks between NS~NF are invalid. F, S, T functions only in G73 are valid.
2.  $\Delta i$ ,  $\Delta k$ ,  $\Delta u$ ,  $\Delta w$  are specified by the same address U, W, and their difference is whether their blocks specify by P, Q.
3. The system cannot call subprograms between NS and NF in G73.
4. NS~NF blocks realize the cycle machining, signs of  $\Delta u$ ,  $\Delta w$ ,  $\Delta i$ ,  $\Delta k$  are paid more attention. The tool returns to A after the cycle is completed.
5. When one of  $\Delta i$ ,  $\Delta k$  is 0, U0 or W0 is compiled in the course of programming; or P73/P74 are set 0, otherwise, it is influenced by last G73's set value.
6. There are most 100 blocks between NS and NF, ERR137 alarm occurs when the blocks exceed 100.

Example Using 73 compiles programs shown in Fig.3-4-3-2.

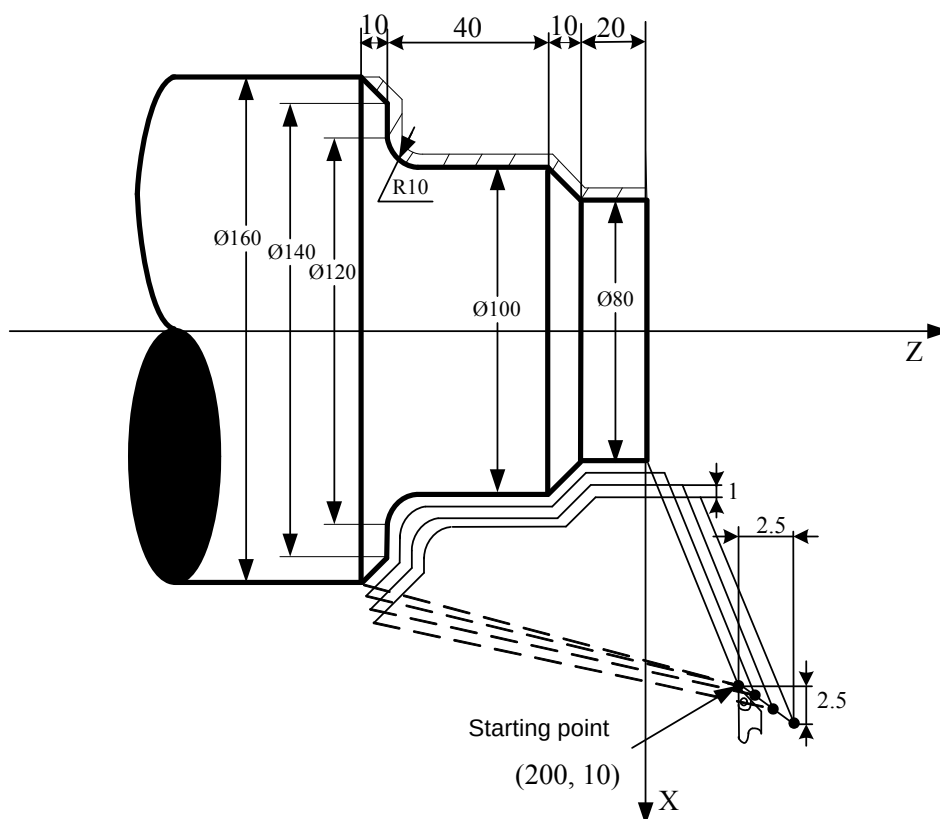


Fig. 3-4-3-2 G73 sample

A program is shown below: (the diameter is designed, metric input, , least blank dimension is Ø86)

```

N008 G0 X260.0 Z50.0 ;           (positioning to safety position)
N009 T0101;                     (execute No. 1 tool, its tool offset)
N010 G98 M03 S300;               (the spindle rotating CW, speed 300)
N011 G00 X200.0 Z10.0;           (rapid position to starting point)
N012 G73 U2.0 W2.0 R3 ;          (X tool retraction 4mm, Z tool retraction 2mm, roughing 3 times,
                                diameter feeding 2mm at every tool infeed)
N013 G73 P014 Q020 U0.5 W0.5 F100 ; (X allowance 0.5mm, Z finishing allowance 0.5mm)
N014 G00 X80.0 W-10.0 S500 ;
N015 G01 W-20.0 F120 ;
N016 X100.0 W-10.0 ;
N017 W-30.0 ;
N018 G02 X120 W-10.0 R10.0 F100 ;
N019 G01 X140.0 ;
N020 G01 X160.0 W-10.0 ;
                                } blocks for finishing shape
N021 G0 X260.0 Z50.0; (traverse to safety position to execute tool change)
N022 T0303;                   (execute No. 3 tool, its tool offset)
N023 G00 X200.0 Z10.0;        (rapid return to G70's positioning)

```

N024 G70 P014 Q020; (finishing)  
 N025 M5 S0 T0200; (stop the spindle, execute No. 2 tool and cancel its tool compensation)  
 N026 G0 X260.0 Z50.0; (rapid return to starting point)  
 N027 M30; (end of program)

### 3.4.4 Finishing Cycle (G70)

**Command format:** G70 P (NS) Q (NF);

**Function:** when G70 is executed, the tool executes the finishing along the workpiece finishing path specified by NS~NF from the initial point.

Using G70 executes finishing after G71, G72, G73 executes roughing.

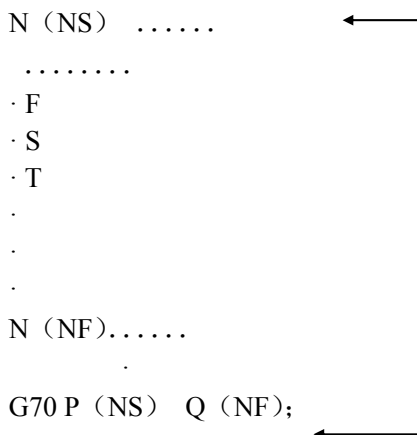
**Explanation:** NS: It is the first block's serial number in blocks for finishing path;

NF: It is the last block's serial number in blocks for finishing path.

G70 path is determined by NS~NF blocks. Relative position of NS, NF in G70~G73 is shown below:

.....  
 .....

G71/G72/G73 P (NS) Q (NF) U ( $\Delta u$ ) W ( $\Delta w$ ) F S T ;



1. F, S, T function among "NS" and "NF" in G71 G72 G73 are invalid, but they are valid in G70.
2. The tool returns to starting point and the system reads the next block when G70 completes.
3. G70 includes S, T, F. In M code, M30 is valid when M30 and G70 are in the same block, and it is invalid when it and other commands are in the same block.
4. Blocks between NS and NF in G70 cannot call subprograms.  
 Example: See G71.G72 sample.
5. The address specified by P.Q should be single and cannot be repetitively commanded. When a program defines the same serial number, the system runs the block for finishing neighboring G70.

### 3.4.5 End Deep Hole Machining Cycle (G74)

**Command format:** G74 R (e);

G74 X (U) Z (W) P ( $\Delta i$ ) Q ( $\Delta k$ ) R ( $\Delta d$ ) F;

**Function:** When G75 is executed, the system confirms the cutting end point according to blocks (the point confirmed by X, Z coordinates)  $e, \Delta i, \Delta k$  and  $\Delta d$  confirms the tool path. In the cycle, it can execute the chip breaking of end cutting. When X (U) are omitted, only Z is executed, which is the deep hole machining. G74 path is shown in Fig. 3-4-5-1.

**Explanation:** e: Tool retraction amount after cutting  $\Delta k$  along Z, unit: mm, range: 0.001mm~9999.999mm.

It is modal and valid before the next is specified. It can be specified by P76, the parameter value also changes according to the program command.

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

U: Difference of X absolute coordinate between cutting end point B2 and starting point A, unit: mm;

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

W: Difference of Z absolute coordinate between cutting end point B2 and starting point A, unit: mm;

$\Delta i$ : X circular movement at one time (without sign, radius value), unit: mm;

$\Delta k$ : Z cutting movement at one time (without sign), unit: mm;

$\Delta d$ : X tool retraction when cutting to end point (radius value), unit: mm;

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

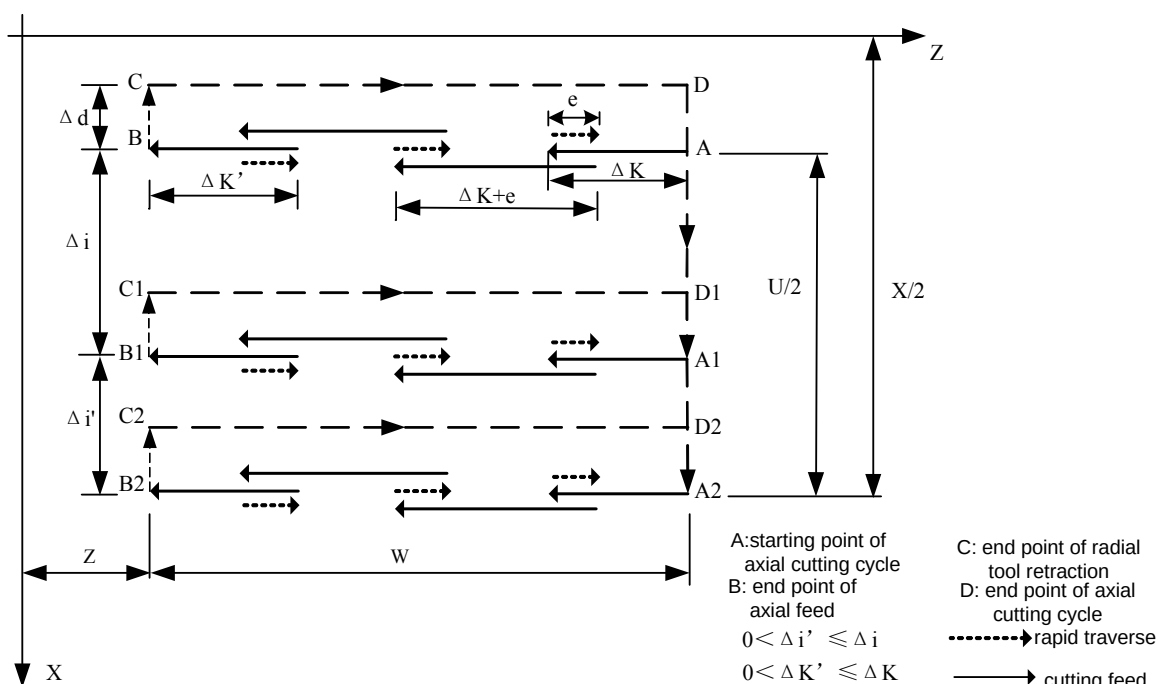


Fig. 3-4-5-1

1. e and  $\Delta d$  are specified by the address R, and their difference are whether they are specified by Z (W), namely, when X (U) is commanded, it is  $\Delta d$ , when X (U) is commanded, it is e;
2. When the cycle operation include G74 of Z (W) and Q ( $\Delta k$ ), "G74 R (e)" is executed, the cycle operation cannot be executed.

Example: Using G74 compiles a program shown Fig. 3-4-5-2.

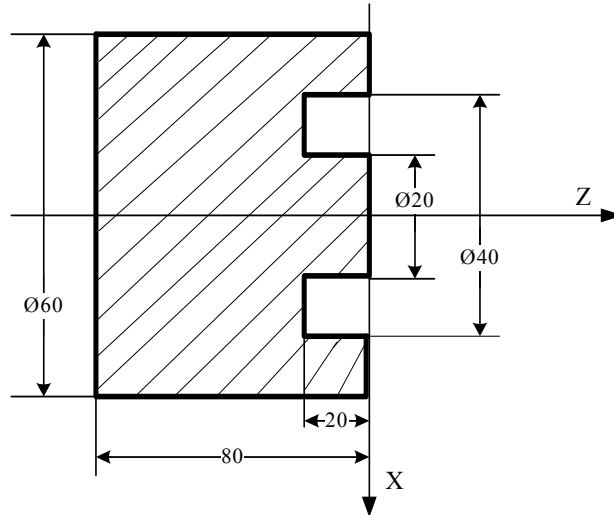


Fig. 3-4-5-2

A program:

```

O0001;           (program name)
G0 X100 Z50;     (rapid positioning )
T0101;          (tool width 2mm)
M3 S500 G97;     (start the spindle, its speed 500)
G0 X36 Z5;       (positioning to starting point of machining, tool width having been added to X
                  value)
G74 R1 ;         (Z tool retraction)
G74 X20 Z-20 P2 Q3.5 F50; (X cycle movement 4mm at one time 4mm, Z 3.5mm)
G0 Z50;          (Z tool retraction)
X100;           (X tool retraction)
M5 S0;           (stop the spindle)
M30;             (end of program)

```

### 3.4.6 Outer Grooving Cycle (G75)

**Command format:** G75 R (e);

G75 X (U) Z (W) P (Δi) Q (Δk) R (Δd) F ;

**Function:** When G75 is executed, the system confirms the cutting end point according to blocks (the point confirmed by X, Z coordinates) e, Δi, Δk and Δd confirms the tool path. It is similar that X, Z are exchanged in G74, and in the cycle, it can execute the chip breaking of end cutting, execute the grooving and cutting-down machining of outer diameter (omit Z, W, Q). G75 path is shown in Fig. 3-4-6-1.

**Explanation:** e: Tool retraction amount after cutting Δi along X, unit: mm, range: 0~9999.999mm. It is modal and valid before the next is specified. It can be specified by P76, the parameter value also changes according to the program command;

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

U: Difference value of X absolute coordinate between cutting end point B2 and A, unit: mm;

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

W: Difference value of Z absolute coordinate between cutting end point B2 and A, unit: mm;  
 $\Delta i$ : X circular movement at one time (without sign, radius value), unit: mm;  
 $\Delta k$ : Z cutting amount at one time (without sign), unit: mm;  
 $\Delta d$ : Z tool retraction amount when cutting to end point, unit: mm;  
F : Cutting feedrate.  
G74, G75 are used to cutting, grooving or hole machining, make the tool automatically retract.

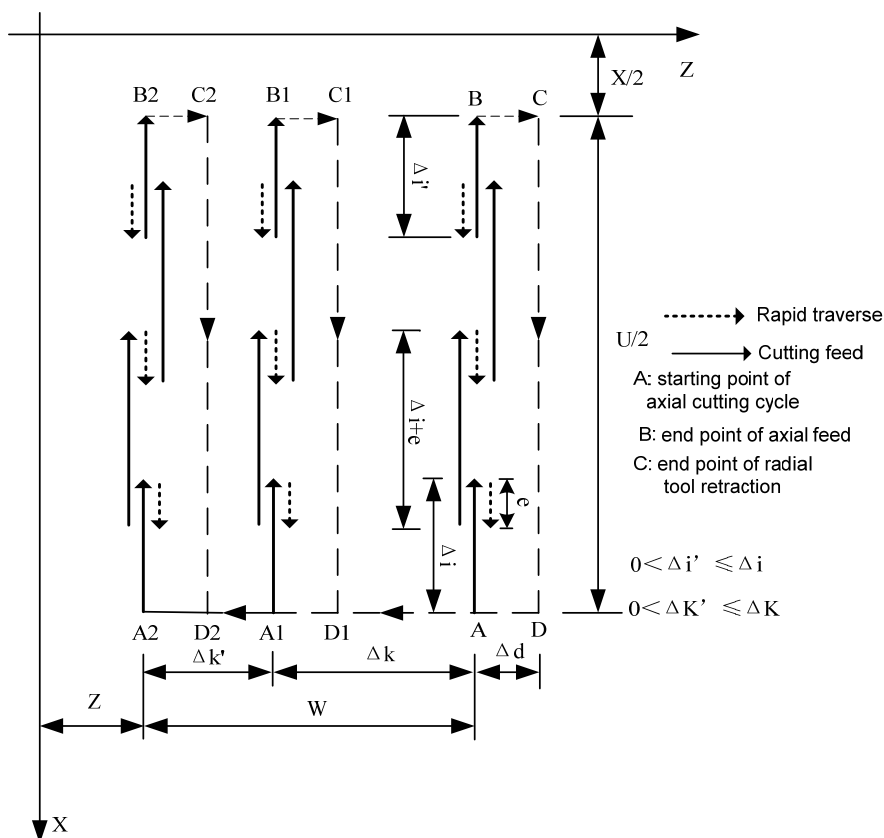


Fig. 3-4-6-1

1.  $e$  and  $\Delta d$  are specified by the address R, and their difference are whether they are specified by X (U), namely, when X (U) is commanded, it is  $\Delta d$ , when X (U) is commanded, it is  $e$ ;
2. The circular operations is executed by G75 specified by X (U).

Example: Using G75 compiles a program shown in Fig. 3-4-6-2.

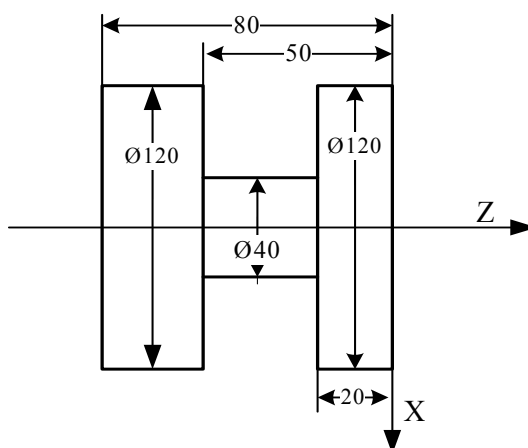


Fig. 3-4-6-2 G75 sample

A program:

```

O0001;          (program name)
G0 X150 Z50;    (rapid positioning)
T0101;          (tool width 4mm)
M3 S500 G97;    (start the spindle, its speed 500)
G0 X125 Z-24;   (positioning to starting point of machining, the tool width having been
                added to Z)
G75 R1 ;        (X tool retraction)
G75 X40 Z-50 P2 Q3.5 F50; (X cycle movement 4mm at one time 4mm, Z 3.5mm)
G0 X150;        (X tool retraction)
Z50;            (Z tool retraction)
M5 S0;          (stop the spindle)
M30;            (end of program)

```

### 3.4.7 Compound Thread Cutting Cycle (G76)

**Command format:** G76 P (m) (r) (a) Q ( $\Delta d_{min}$ ) R (d);  
 G76 X (U) Z (W) R (i) P (k) Q ( $\Delta d$ ) F (I) ;

**Function:** The system automatically counts and executes thread cutting cycle many times according to the commanded address, and G76 path is shown in Fig. 3-4-7-1.

**Explanation:** X.Z: coordinates of thread end point (bottom of thread), unit: mm;

U.W: coordinates of thread end point corresponding to machining starting point, unit: mm;

m: it is repetitive times of last finishing, modal, valid before it is specified next time. It is set by P77, the parameter value is also changed according to the commands. Range: 1~99;

r: Chamfer value of thread. When L is the lead, in  $0.1L \sim 9.9L$ , 0.1L is taken as the first gear, using 00~99 two numerical values is specified. It is modal and valid before it is specified next time. Using P68 also change the parameter value according to the commands. After G76 set the thread chamfering value, it influences in G92 thread cutting cycle.

a: Tool nose angle (angle of thread tooth can select the angle  $80^\circ$ ,  $60^\circ$ ,  $55^\circ$ ,  $30^\circ$ ,  $29^\circ$ ,  $0^\circ$ ). The original of angle value is specified by two numerical value. It is modal, and valid before it is specified next time. It is set by P78, and the parameter value is changed according to the command. The tool nose angle can select  $80^\circ.60^\circ.55^\circ.30^\circ.29^\circ.0^\circ$ ;

$\Delta d_{min}$ : least cut-in value, unit: mm. When the first cut-in value ( $\Delta D \times \sqrt{N} - \Delta D \times \sqrt{N-1}$ ) is less than  $\Delta d_{min}$ ,  $\Delta d_{min}$  is taken as the first cut-in value. It is modal, and valid before it is specified next time. It is also set by P79, the parameter value can be changed by the commands. The least cut-in value is 0~9999999, unit: 0.001mm;

d: Finishing allowance, unit: mm. It is modal, and valid before it is specified next time. It is set by P80, the parameter value is changed by the command. The finishing allowance range is 0~9999999, unit: 0.001mm;

i: Radius difference of thread part, unit: mm, i=0 : cutting the straight thread;

k: Tooth height of thread (X distance is commanded by the radius value), unit: mm;

$\Delta d$ : The first cutting depth, radius value, unit: mm.

F: Thread lead, unit: mm;

I: Tooth/inch.

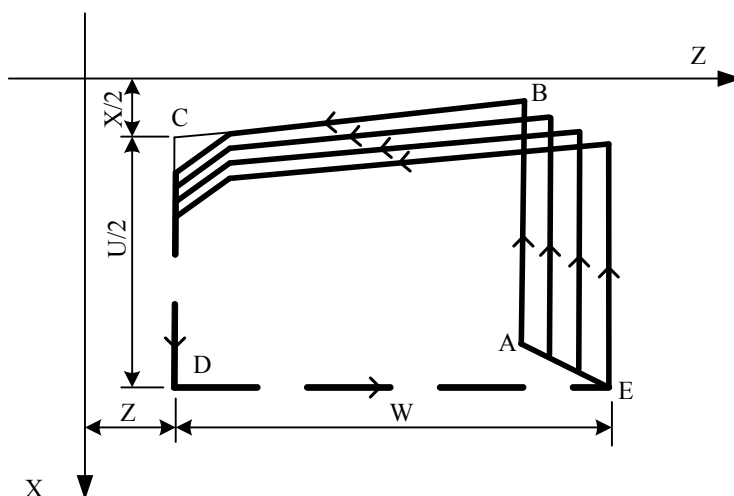


Fig. 3-4-7-1

Cut-in method is shown in Fig. 3-4-7-2:

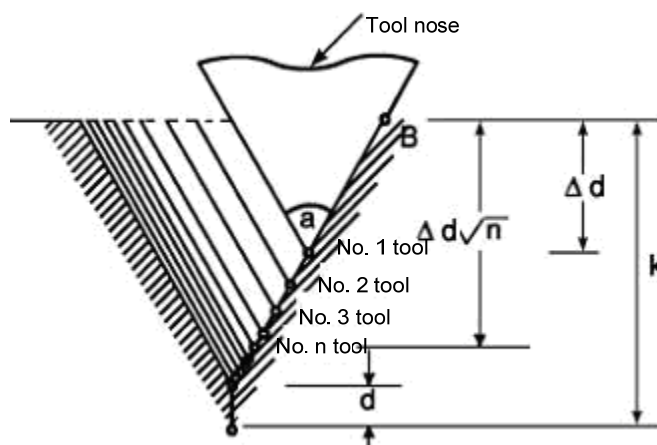


Fig. 3-4-7-2

1. Data specified by P, Q, R are distinguished by whether they have X (U), Z (W) or not.
2. The cycle operation is executed by G76 specified by the address X (U), Z (W) .
3. In cycle machining, the tool uses the one-side to machine, and the tool nose load can be reduced.
4. The first cut-in amount is  $\Delta d$ , the N times cut-in is  $\Delta d \times \sqrt{N}$ , each cut-in amount is fixed.
5. There are four machining shapes, which can machine inside thread. B, C section uses the feedrate commanded by F, others are rapid traverse in Fig. 3-4-7-1.

Increment signs are determined as follows:

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

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

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

P (K): positive;

Q ( $\Delta D$ ): positive.

6. Its notes are the same those of G32.
7. The thread chamfering value designation is valid to G92 thread cutting cycle.
8. m, r, a are specified by the address p one time.

Example: using G76 compile a program shown in Fig. 3-4-7-3, machining thread is M68X6.

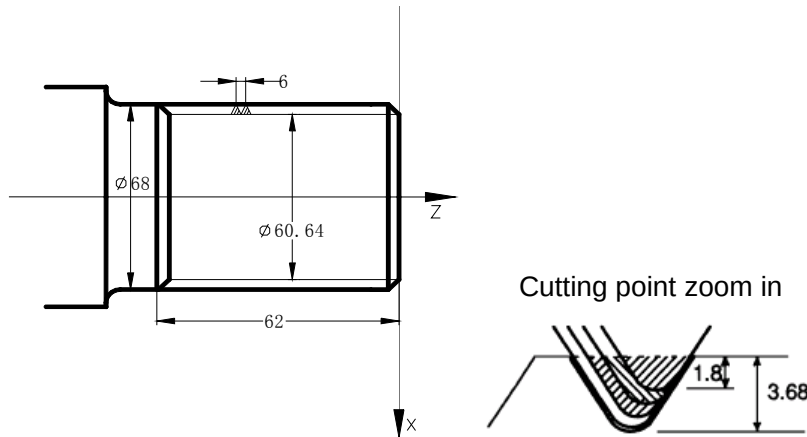


Fig. 3-4-7-3

A program:

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

### 3.4.8 Notes of Compound Fixed Cycle Codes

1. There must be P, Q, X, Z, U, W, R in blocks of the commanded compound fixed cycle, each block must be commanded correctly.
2. In blocks of G71, G72, G73, when their serial numbers are specified by P, the blocks of corresponding serial number must command G00 or G01 in Group 01, otherwise, P/S alarm occurs.
3. In MDI mode, the system cannot execute G70, G71, G72, G73, G74, G75, G76. Namely, they cannot be executed even if they are commanded.
4. In G70, G71, G72, G73, the blocks which serial numbers are specified by P and Q cannot have the following Codes:
  - ★ Codes in Group 01 except for G00, G01, G02, G03;
  - ★ M98/M99;
  - ★ G04 is valid in the last forming tool of roughing and valid in finishing.
5. When the compound fixed cycle (G70~G76) is executed, the operation can be stopped to insert the manual operation, but the system must return the previous operation before the compound fixed cycle is executed again. If not, the manual movement amount will not be added to the absolute value, the following operation will miss position, which value is equal to manual movement amount.

6. When G70, G71, G72, G73 is executed, the serial numbers specified by P, Q cannot be repetitive in the program.
7. Notes of G76 are same those of G32, G92, are valid for thread chamfering designation and G92.

## Chapter Four Miscellaneous Function M Codes

The movement command and M code must be executed simultaneously when they are in the same block.

(Example ) N1 G01 X50.0 Z-50.0 M05 ; (spindle stop)

### 4.1 Miscellaneous Function (M function)

Miscellaneous function M code consisted of the address M and two-digit numerical value. The system sends the corresponding control signal to the machine, used to control the corresponding function ON/OFF of the machine. There must be only one M code in the same block.

The system supports the following M codes:

|           |                                                                                                                      |
|-----------|----------------------------------------------------------------------------------------------------------------------|
| M03 :     | Spindle rotation CW.                                                                                                 |
| M04 :     | Spindle rotation CCW.                                                                                                |
| M05 :     | Spindle stop.                                                                                                        |
| M08 :     | Cooling ON.                                                                                                          |
| M09 :     | Cooling OFF (do not output signals) .                                                                                |
| M10 :     | Tailstock forward.                                                                                                   |
| M11 :     | Tailstock backward.                                                                                                  |
| M12 :     | Chuck clamping.                                                                                                      |
| M13 :     | Chuck releasing.                                                                                                     |
| M32 :     | Lubricating ON.                                                                                                      |
| M33 :     | Lubricating OFF (do not output signals) .                                                                            |
| M00 :     | Program pause; press 'Cycle Start' key to continuously execute programs.                                             |
| M30 :     | End of program; the program returns to the initial.                                                                  |
| M41~M44 : | Automatic gear-change function of spindle (see Machine Connection Section, Automatic Gear-Change Control of Spindle) |
| M51~M70 : | User-customized output control                                                                                       |
| M91~M94 : | User-customized input control                                                                                        |

Except for M00, M30, execution time (not pulse width) of other M codes is set by P86.

Set value: 0~9999999

Set time = set value × 4 ms.

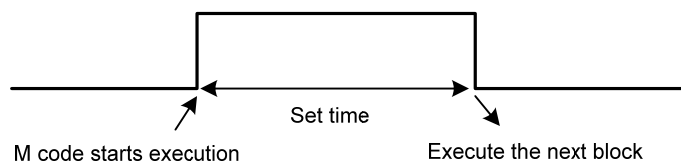


Fig. 4-1-1

**Note 1:** When other M codes except for the above, the system alarms the following and stops the execution.

**NO.181:** M code is wrong, illegal M codes are input into the program.

**Note 2:** After M, S, T, function is activated, the mode still remains even if it is changed, 'Reset' key is pressed to close the activation. (NO:12#0 sets whether M, S, T function is valid.)

The following M codes describe special usages:

1. M30 end of program
  - 1) M30 means the end of main program.
  - 2) It means the system stops the automatic run, is in reset state.
  - 3) It means the system returns to the beginning of the main program.
  - 4) It means 1 is added to the number of machining workpiece.

2.M00: program pause

After the block of M00 is executed, the system stops the automatic run. It is the same that of the single block stop, and the system saves the previous modal information. The system starts the automatic run after it runs.

3.M98/M99 (call subprogram/ subprogram return)

**Format:**



It can be execute repetitively when it is used for calling a subprogram or the end of program is M99. The four-embedding of subprogram can be executed and is referred to Section 4.7 Subprogram Control.

**Note 1:** The next block of M00, M30 cannot be saved into the buffer storage even if it exists.

**Note 2:** The code signals are not output when M98, M99 is executed.

## 4.2 Special M Codes (M21.M22.M23.M24 )

Relevant Parameters:

|   |   |   |  |  |     |     |  |  |  |  |
|---|---|---|--|--|-----|-----|--|--|--|--|
| 0 | 1 | 3 |  |  | M23 | M21 |  |  |  |  |
|---|---|---|--|--|-----|-----|--|--|--|--|

**M21** =0: the corresponding output pin does not execute the output

=1: the corresponding output pin executes the output

**M23** =0: the corresponding output pin does not execute the output

=1: the corresponding output pin executes the output

### Input/output signals:

M21I : input signal (user-customized input pin) M21O: output signal (user-customized output pin)

M23I : input signal (user-customized input pin) M23O: output signal (user-customized output pin)

### Use method:

Function 1: M21 When M21=0, the corresponding output pin does not execute the output; when M21=1, it outputs M21O.

Example: G0 X100 Z100

M21 execute the normal M code

G0 X0 Z0

M22 close output M21O

M30

**Function 2:** M21 P when M21 is set 1 and after M21 P function is executed in the time specified by P, M21 output ends.

Example: G0 X100 Z100  
**M21** P10000 the duration of M21 execution is 10s, and M21O is closed.  
 G0 X0 Z0  
 M22  
 M30

**Function 3:** M21Q When M21 is set 1 and M21 Q function is executed, the corresponding output pin has the output signal and waits for the input signal, and there is an input signal, the system waits for the end to execute the next block; when there is no input signal, the system always waits.

Example: G0 X100 Z100  
 M21 Q1 the system checks input interface M21I, and waits always without it.  
 G0 X0 Z0  
 M22  
 M30  
 M22 close the output M21O.

**Note 1:** The time unit is ms when P is specified.

**Note 2:** When Q is specified, its numerical value is not 0, it is HIGH, otherwise, it is LOW.

**Note 3:** M23 use methods are the same those of M21; M24 of M22.

### 4.3 M Code Calling Subprograms

When NO.4#2(CM98) is set to 1 and the system executes others except for M, S, T, the system calls a corresponding subprogram instead of alarm. The user can extend miscellaneous function codes according to requirements combined with macro input/output interface variable.

M code: when the system executes others except for standard codes, the called subprograms are :

**M□□:** called subprogram **90□□**.

### 4.4 S Codes Calling Subprograms

When NO.4#2(CM98) is set to 1 and the system executes others except for M, S, T, the system calls a corresponding subprogram instead of alarm. The user can extend miscellaneous function codes according to requirements combined with macro input/output interface variable

S code: when the system executes **S10~S99**, the called subprograms are:

**S□□:** called subprogram **91□□**.

**Note 1:** S codes cannot call subprograms when the spindle analog voltage output is selected.

**Note 2:** When the system executes the non-standard M, S, the corresponding subprograms must be input otherwise, NO.78 alarm occurs.

**Note 3:** The non-standard M, S, T codes cannot run in MDI mode, otherwise, M/S or T code wrongly alarms or NO.130 alarm occurs.

**Note 4:** In the corresponding subprogram, the axis motion command can be input, or the output point can be controlled (ON/OFF), or the skip or cycle is executed, some DI signal is taken as an end signal of M/S/T. See Macro Program about DI/DO.

#### 4.5 T Codes Calling Subprograms

When NO.4#2(CM98) is set to 1 and the system executes others except for M, S, T, the system calls a corresponding subprogram instead of alarm. The user can extend miscellaneous function codes according to requirements combined with macro input/output interface variable.

T code: when the system executes others except for standard codes, the called subprograms are :

T□□: called subprogram 92□□.

For example: T28 calls the subprogram 9228.

#### 4.6 Parameters of Miscellaneous Function

The parameters are set by the user according to the actual.

Setting: Open the parameter protection switch, select the parameter window in MDI mode, move cursor

to the front of the required serial number to input data, press  and then the data is input into the corresponding parameters.

Methods for searching a serial number are shown below:

- 1) Use Page Up/Down to find out the serial number window, move the cursor by the direction key to position to the required serial number position.

- 2) Search method: press  →input the required searched parameter number →press



#### 4.7 Alarm Related to Miscellaneous Function

Alarms related to miscellaneous function are displayed on the external message window. When the system alarms, it automatically switches to external message window, and the detailed contents of alarm are displayed on the top of LCD, 'ALARM' flashed at the right of LCD.

181: M code error.

Illegal M codes are input into the program.

182: S code error.

Illegal S codes are input into the program.

183: T code error.

Illegal T codes are input into the program.

185: The tool change time is too long.

The system alarms when it has not received the specified cutter spacing arrival signal after the time of the tool post rotating forward for Ta .

186: The system has not received the tool post's reverse locking signal within the tool post reverse locking time.

187 : Tool change is not ready.

The system does not check whether the tool number is consistent with the current tool number.

044: Start the spindle when the chuck clamps.

045: The chuck releases when the spindle runs.

250: The tailstock advancing/retreating is executed Automatic run, starting the spindle, the spindle rotating.

## Chapter Five Spindle Function S Code

The code signal is sent to machine by the address S and its following numerical value, used to control the spindle of the machine. In a block, one S code can be commanded.

Refer to the manufacturer's user manual about digits and use of S code.

When the movement command and S code are in the same block, they are executed simultaneously.

### 5.1 Spindle Switching Value Control

NO:1#4=0: the address S and its following numerical value control the spindle's speed.

**Command format:** S\_\_

The system provides 4-gear spindle mechanic gear change. Refer to the manufacturer's user manual about the corresponding relation between S code and the spindle speed, and the spindle speed gear of the machine (without selecting the analog spindle function).

Gear signal: S1~S4

Duration of S code is set by P87.

Set value: 0~9999999

Set time = set value  $\times$  4 ms.

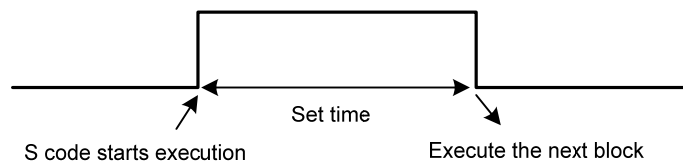


Fig. 5-1-1

**Note:** The system alarms below and stops the operation when the program specifies others except for the above S codes.

**182:** S code error, illegal S code is input to the program.

### 5.2 Spindle Analog Value Control

NO:1#4=1: the address S and its following numerical value directly commands the rotation quantity of the spindle (r/min), the unit of quantity of the spindle is different because of the different machine manufacturer. P55~P58 sets the max. speed of the spindle.

**Command format:** S\_\_

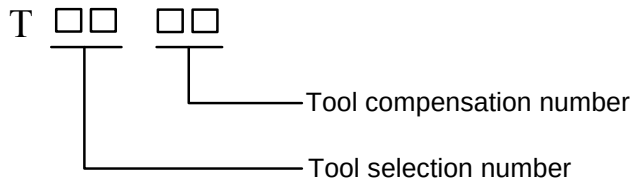
The system realizes the spindle stepless speed regulating when the spindle analog value control is activated.

## Chapter Six Tool Function T Code

Use the address T and its following 2-digit numerical value to select the tool installed on the machine. In one block, the system can command a T code. They are executed simultaneously when the movement command and T code are in the same block.

Refer to the machine manufacturer's user manual about T code use.

Use T code and its following numerical value to select the tool. The following two-digit numerical value are used for specifying the compensation number of the tool compensation.



The tool quantity provided by the system is set by P81, and its most setting is 8.

### 6.1 Tool change Process

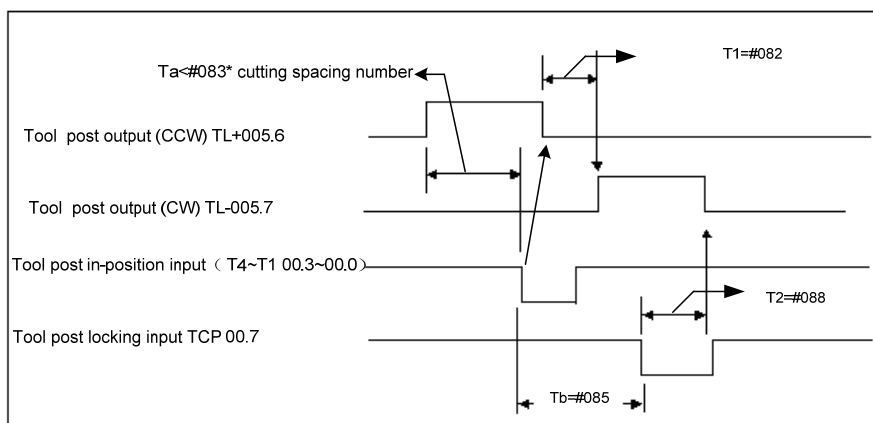


Fig. 6-1-1

In the above figure, P83 setting corresponds to the time parameter set by the parameter number.

When  $Ta \geq P8 \times$  quantity of tool change, NO.185 alarm occurs: the tool change time is too long. The system alarms when it has not received the specified tool arrival signal after the time of the tool post rotating forward for  $Ta$ .

When  $Ta \geq P85 \times$  quantity of tool change, NO.186 alarm occurs: the tool change time is too long. The system alarms when it has not received the specified tool arrival signal after the time of the tool post rotating reversely.

When T code is executed, the system output the tool post rotation (forward rotation signal) (TL+), and after it has received the tool arrival signal specified by T code, closes the signal and delays T1, then the tool post starts the reverse rotation to lock (TL-) and checks the locking signal \*TCP; and after it has received the locking signal, delay the time set by P85, closes the reverse signal (TL-). When the executed tool number and the current tool number (it is recorded automatically in P49), the tool change command stops and the system executes the next block, otherwise, NO.187 alarm occurs.

After the system outputs the tool post's reverse rotation signal, has not received \*TCP signal in the time set by P88, it alarms and closes the rotation signal.

**Note:** When the tool post mask position offsets, NO.187 alarm occurs after tool change, at the moment, the user should regulate the mask position or set No.10 BIT3 to 0.

## 6.2 Relevant Parameters of Tool Change

### 1) System parameters

The tool post in-position signal (\*T8~\*T1) HIGH or LOW is set by **NO:9#1 TSGN**.

**TSGN 0 :** the tool post in-position signal HIGH is valid. (normally-closed)

**1 :** the tool post in-position signal LOW is valid. (normal open)

The tool post locking signal (\*TCP) HIGH or LOW is set by **NO:9#0 TCPS**.

**TCPS 0 :** the tool post locking signal LOW is valid. (normal open)

**1 :** the tool post locking signal HIGH is valid. (normally-closed)

**Note:** When there is no tool post locking signal, it cannot be connected, it is valid when TCPS is LOW, P085 sets the reverse locking time. 980TB3 has no the signal \*TCP, and TCPS is set to "0".

### 2) Relevant parameters of tool post

**T1 :** delay time from the tool post stopping forward to starting reversely rotation.

P82, set value: 0~9999999 unit: 4ms

**Ttool quantity :** tool quantity of tool post.

P81, set value 1~8 unit: piece.

**T2 :** delay time after receiving locking signal of tool post reversely rotation.

P85, set value 0~9999999 unit: 4ms

**Tone tool:** the maximum time of tool change.

P83, set value 0~9999999 unit: 4ms

**T total tools:** the maximum time from the first to the last tool change.

P84, set value 0~9999999 unit: 4ms

### 3) Diagnosis parameters

**Tcurrent tool number:** current tool number.

The diagnosis number P49. The system automatically sets its value, namely, the manual tool change is executed, i.e. automatic setting when the value is used firstly or the storage is cleared.

**Ta :** The system automatically count the required maximum time from current tool change to the specified tool.

**Ta = T tool** × tool quantity of tool change.

**Example:** Supposing that the total tool quantity is 6.

a) The current tool is **1**, the specified is **5**, the tool of tool change is **4**.

b) The current tool is **5**, the specified is **2**, the tool of tool change is **3**.

**Tb :** The longest time signal for having received the tool post reverse rotation signal

## 6.3 Tool Offset

In the actual machining, the reference tool in programming is not almost consistent with the actual

machining tool, at the moment, the difference between the standard position and actual tool nose is called offset value.

The tool offset is controlled by T code instead of G code.

### 6.3.1 Basic Tool Offset

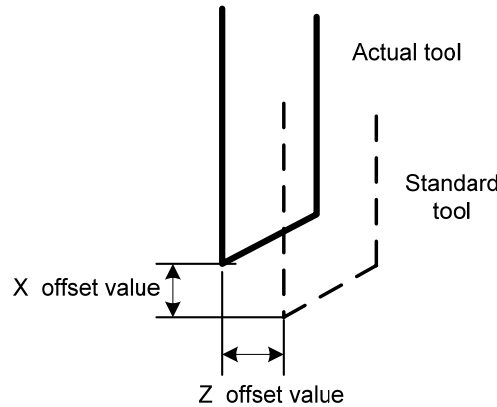
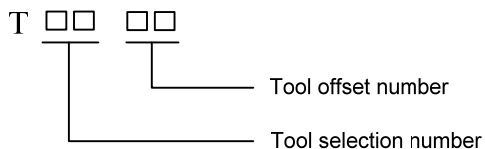


Fig. 6-3-1-1

The offset result makes the tool nose of the standard tool move in the programmed path, the tool installation is consistent with the initial point of standard tool in program, but the actual machining tool is not almost consistent with the standard. The difference between the standard position and the actual tool nose position is called offset value.

### 6.3.2 T Code for tool offset

T code's meanings:



#### a) tool selection

The tool selection is executed by specifying T code corresponded to the tool number.

Refer to the machine manufacturer's user manual about relationship between the tool selection number and the tool

#### b) tool offset number

It is used for selecting the offset value corresponded to the offset number. The offset value must be input by keyboard. The corresponding offset number has two offset values, one for X and another for Z. See Tool Compensation Display, Modification and Setting about its concrete operations.

Table 6-3-2-1

| Offset value  |                |                |
|---------------|----------------|----------------|
| Offset number | X offset value | Z offset value |
| 01            | 0.040          | 0.020          |
| 02            | 0.060          | 0.030          |
| 03            | 0              | 0              |
| ..            | .              | .              |
| ..            | .              | .              |
| ..            | .              | .              |

The tool offset is valid when T code is specified and its offset number is not 00.

When the offset number is 00, the tool offset function is cancelled.

Offset value is set:

Input in mm: -9999.999 mm~9999.999mm

**NO:2#5 ORC** can specify the diameter/radius specifications to X tool offset value.

### 6.3.3 Offset

Offset X, Z is for programmed path. T code specifies the offset value of offset number, and the offset value is added or subtracted at the end point of each block.

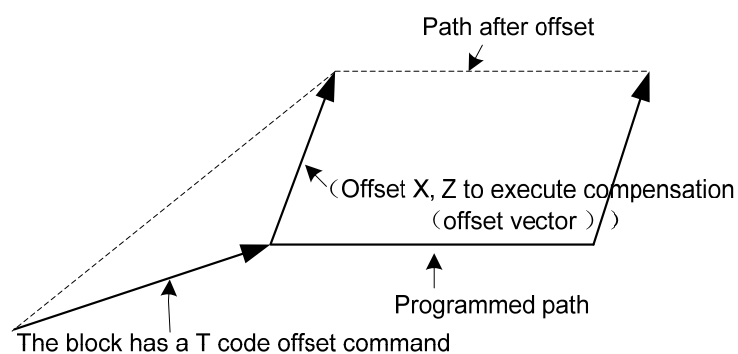


Fig. 6-3-3-1

#### a) Offset vector

In above figure, X, Z vector with offset is called offset vector. The compensation affects the offset vector.

#### b) Offset cancellation

When the offset number of T code selects 00, the offset is cancelled. The offset vector is 0 at the end of canceling blocks.

N1 G01 U50.0 W100.0 T0202;

N2 W100.0;

N3 U0.0 W50.0 T0200;

Offset path

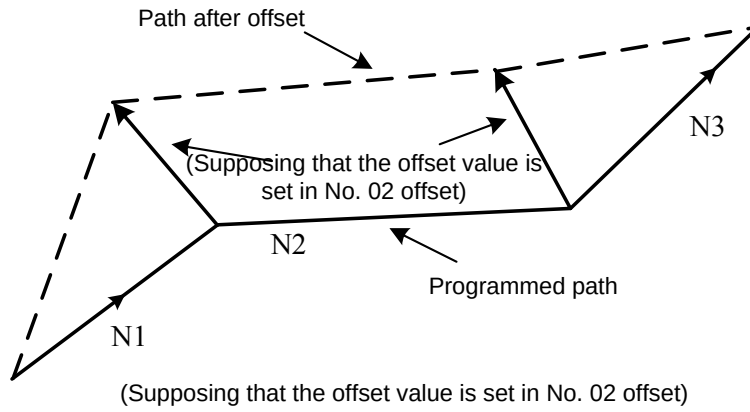


Fig. 6-3-3-2

### 6.3.4 Programming Example

| Tool nose offset (Z, X)       | tool number |
|-------------------------------|-------------|
| Tool #1.....B (0.120, 0.200)  | 01          |
| Tool #2.....C(-0.180, -0.050) | 02          |

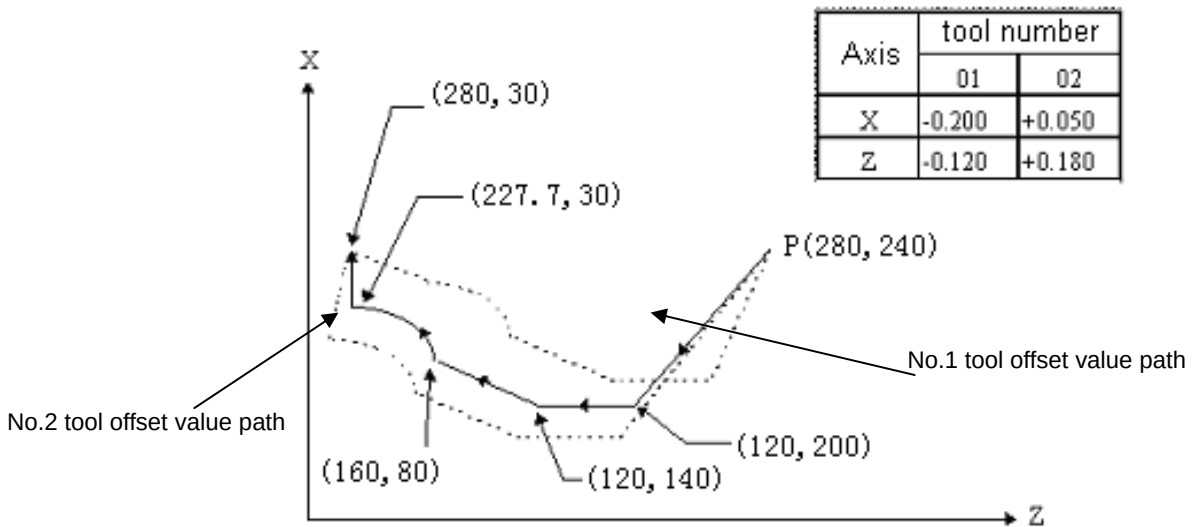


Fig. 6-3-4-1 tool compensation example- 2

(Program example 1)

```
G00 X280.0 Z240.0;
G00 X120.0 Z200.0 T0101;
G01 Z140.0 F30;
X160.0 Z80.0;
G03 X227.7 Z30.0 R53.81;
G00 X280.0 T0100;
```

#1 tool nose path is the same that of the programmed of the program.

(Program example 2)

Modifying example 1, #2 tool nose path is the same that of the programmed.

T0101→T0202 and T0100→T0200

### 6.3.5 Single T code

When T code and the movement function command are in the same block, the traverse speed to execute the tool length compensation is decided by the movement command to cutting feedrate or rapid traverse speed.

**NO:2#0=0:** when a single T code is executed, the speed of tool length compensation is determined by the current mode: when the current mode is the cutting feed, the tool length compensation is executed at the current cutting feedrate; when the current mode is G00, it is done at the current rapid traverse speed.

**NO:2#0=1:** when a single T code is executed, the tool length compensation and the first subsequent movement command are executed simultaneously, the speed of the tool length compensation is determined by the movement command.

## Chapter Seven User Macro Program

A group of command for realizing some function is saved to storage in advance like subprograms, one command standing for these functions is used. Only the command input in programs can realize these function. The group is called user macro program body, the command is called 'user macro command'. The user macro program body is called macro program sometime. The user macro code is also called macro program call command.

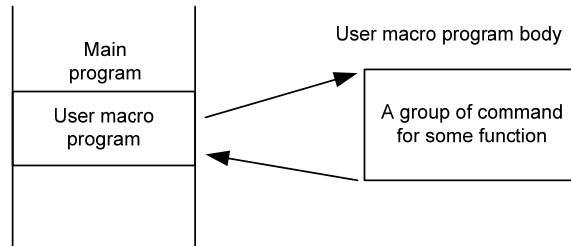


Fig. 7-1

The programmer need not remember macro program body, but does user macro codes as commands. Obvious features of user macro programs the variable can be used in user macro program body. The variables can be operated and macro code is assigned.

### 7.1 User Macro Code

The user macro code is a command to call user macro program body.

Command format:

M98 P□□□□

Program number of called macro program body

Using the above command can call macro program body specified by P.

### 7.2 User Macro Program Body

The user macro program body can use generally CNC command, or use variable, operation or transfer command.

User macro program body begins with a program number following O and ends with M99.

|                   |                                |
|-------------------|--------------------------------|
| O08000;           | Program number                 |
| G65 H01 ... ..;   | Operation command              |
| G00 X#101 ... ..; | CNC command using variable     |
| ...;              |                                |
| ...;              |                                |
| G65 H82 ... ..;   | Transfer command               |
| M99;              | End of user macro program body |
| ...;              |                                |
| ...;              |                                |
| M99;              |                                |

Fig.7-2-1 User macro program body constitution

## 1) Variable usage

Variables can command the address values in user macro program body. The variable value is assigned by a main program or set by key board, or is assigned when the user macro program body is executed. Use many variables, and they can be distinguished by their variable numbers.

## a) Variable meaning

The variable number with # means the measurement, and its format is shown below:

# (i=200, 202, 203, 204.....)

(Example) #205, #209, #1005

## b) Variable reference

Using variable can replace the numerical value following the address.

When a program has “< Address > # i” or “< Address > -# i”, it means to take the variable value or the negative of the variable value as the address value.

(Example) F#203...#203=15: it is the same as F15.

Z-#210...#210=250: it is the same as Z-250.

G#230...#230=3: it is the same as G3.

Variable replacing its variable number can describe into ##9200 instead of #9200, “9” following # means the replacement variable number.

The following examples are to displace the variable number.

(Example) #200 = 205, #205 = 500:

X#200 and X500 are the same.

X-#200 and X-500 are the same.

**Note 1:** O and N cannot reference variables, and cannot use O#200, N#220 in programming.

**Note 2:** When the variable exceeds the maximum programmable dimension described by the address, it cannot be used. When #230 = 120, M#230 exceeds its maximum programmable dimension.

**Note 3:** Display and setting of variable value: it can be displayed on LCD window, and can be set by press key.

## 2) Variable classification

The variable is divided into the common and system according to the variable number, and their use and properties are different.

## a) Common variable #200~#231

The common variable is common in main programs or user macro programs called by main programs. Namely, variable # i used in some user macro program and # i used in other macro

programs are the same. So, the common variable #i of operation result in some macro program can be used to other macro programs.

The common variable use is defined by the user.

Common variable #200~#231 are all "0" when the power supply is turned on again, after the power supply is turned on again.

b) Common variable #500~#515

The variable use is the same those of #200~#231. After the power supply is tuned off, the variable data is saved, and remains when power-on again.

c) System variable #1000~#1031

The variable use is fixed in the system.

Interface input signals #1000~#1031(select the function- need to match corresponding selected parts).

After the system reads the system variable #1000~#1031 values as the interface signals, it can learn states of interface input signals.

Diagnosis No.

001

Programmed  
variable number

| Tool No.: 7       | 6                 | 5                | 4                | 3                | 2                | 1                | 0                 |
|-------------------|-------------------|------------------|------------------|------------------|------------------|------------------|-------------------|
| TCP               | DIQP              | *DECX            | DITW             | *SP              | *ST              | *DEC<br>Z        | *ESP              |
| #1007<br>XS39: 12 | #1006<br>XS39: 11 | #1005<br>XS40: 1 | #1004<br>XS40: 2 | #1003<br>XS40: 7 | #1002<br>XS40: 8 | #1001<br>XS40: 9 | #1000<br>XS40: 10 |

Pin

| T08               | T07               | T06               | T05               | T04              | T03              | T02              | T01              |
|-------------------|-------------------|-------------------|-------------------|------------------|------------------|------------------|------------------|
| #1015<br>XS40: 19 | #1014<br>XS40: 20 | #1013<br>XS40: 21 | #1012<br>XS40: 22 | #1011<br>XS40: 3 | #1010<br>XS40: 4 | #1009<br>XS40: 5 | #1008<br>XS40: 6 |

003

| SAR              | LTZ               | LTX              | SPEN             | PCH              | DOOR             | GR2              | GR1              |
|------------------|-------------------|------------------|------------------|------------------|------------------|------------------|------------------|
| #1023<br>XS41: 7 | #1022<br>XS41: 19 | #1021<br>XS41: 6 | #1020<br>XS41: 5 | #1019<br>XS41: 4 | #1018<br>XS41: 3 | #1017<br>XS41: 2 | #1016<br>XS41: 1 |

004

| M01               | M93               | M91                | LTY               | DECY             | LCK               | OWA2             | OWA1              |
|-------------------|-------------------|--------------------|-------------------|------------------|-------------------|------------------|-------------------|
| #1031<br>XS42: 12 | #1030<br>XS42: 11 | #1029<br>XS41 : 10 | #1028<br>XS41: 22 | #1027<br>XS41: 9 | #1026<br>XS41: 21 | #1025<br>XS41: 8 | #1024<br>XS41: 20 |

Value of 32-digit DI signal is automatically assigned to macro variable (#1000~#1031) inside the system. It with other logic judge, skip macro code are used together to perform various of execution.

#1000~1031 are values of corresponding points (0/1).

Interface output signal #1100~1131 (select the function- need to match corresponding selected parts).

Assign the system variable #1100~1131 to change state of output signals.

|                         |           |          |          |          |          |          |          |         |
|-------------------------|-----------|----------|----------|----------|----------|----------|----------|---------|
| Diagnosis No.           | Bit No: 7 | 6        | 5        | 4        | 3        | 2        | 1        | 0       |
| 005                     | SPZD      | DOQPJ    | M05      | M32      | M08      | M10      | M04      | M03     |
| Macro program           | #1107     | #1106    | #1105    | #1104    | #1103    | #1102    | #1101    | #1100   |
| Programmed variable No. | XS40: 17  | XS40: 4  | XS39: 16 | XS39: 6  | XS39: 15 | XS39: 2  | XS39: 3  | XS39: 7 |
| Pin                     |           |          |          |          |          |          |          |         |
| 006                     | TL-       | TL+      | DOQPS    | M11      | M44      | M43      | M42      | M41     |
|                         | #1115     | #1114    | #1113    | #1112    | #1111    | #1110    | #1109    | #1108   |
|                         | XS40: 13  | XS40: 12 | XS39: 10 | XS39: 9  | XS39: 8  | XS39: 14 | XS39: 1  | XS39: 5 |
|                         | M53       | M51      | TLC      | STAR     | TRIAN    | WAR      | M30      | MST     |
|                         | #1123     | #1122    | #1121    | #1120    | #1119    | #1118    | #1117    | #1116   |
|                         | XS42: 17  | XS42: 4  | XS42: 16 | XS42: 15 | XS42: 3  | XS42: 2  | XS42: 14 | XS42: 1 |
| 008                     | M69       | M67      | M65      | M63      | M61      | M59      | M57      | M55     |
|                         | #1131     | #1130    | #1129    | #1128    | #1127    | #1126    | #1125    | #1124   |
|                         | XS41: 13  | XS41: 12 | XS42: 10 | XS42: 9  | XS42: 8  | XS42: 7  | XS42: 6  | XS42: 5 |

32-digit DO signal outputs 1 or 0 by macro variable (#1100~#1131) assignment.

### 7.3 Operation Command and Transfer Command (G65)

General format:

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

m: 01~99 means operation command or transfer command function.

# i: variable name to save operation result.

# j: variable name 1 to execute operation, it can be constant.

# k: variable name 2 to execute operation, it can be constant.

Meaning: # i = # j O # k

\_\_\_\_\_ It is an operator, is specified by Hm.

(Example) P#200 Q#201 R#202.....#200 = #201 O #202;

P#200 Q#201 R15....#200 = #201 O 15;

P#200 Q-100 R#202.....#200 = -100 O #202;

**Note 1:** The variable value does not include the decimal point. Meanings of each variable value and each address without decimal point are the same.

(Example) #200 = 10

X#200=X 10mm(input with mm)

**Note 2:** The constant is directly described without #.

**Note 3:** H code specified by G65 does not affect the offset value.

Table 7-3-1

| G code | H code | Function                                     | Definition                                          |
|--------|--------|----------------------------------------------|-----------------------------------------------------|
| G65    | H01    | Assignment                                   | $\#i = \#j$                                         |
| G65    | H02    | Addition                                     | $\#i = \#j + \#k$                                   |
| G65    | H03    | Subtraction                                  | $\#i = \#j - \#k$                                   |
| G65    | H04    | Multiplication                               | $\#i = \#j \times \#k$                              |
| G65    | H05    | Division                                     | $\#i = \#j \div \#k$                                |
| G65    | H11    | Logic addition (OR)                          | $\#i = \#j \text{ OR } \#k$                         |
| G65    | H12    | Logic multiplication (AND)                   | $\#i = \#j \text{ AND } \#k$                        |
| G65    | H13    | OR                                           | $\#i = \#j \text{ XOR } \#k$                        |
| G65    | H21    | Square root                                  | $\#i = \sqrt{\#j}$                                  |
| G65    | H22    | Absolute value                               | $\#i =  \#j $                                       |
| G65    | H23    | Remainder                                    | $\#i = \#j - \text{trunc}(\#j \div \#k) \times \#k$ |
| G65    | H24    | Decimal to binary system                     | $\#i = \text{BIN}(\#j)$                             |
| G65    | H25    | Binary to decimal system                     | $\#i = \text{BCD}(\#j)$                             |
| G65    | H26    | Composite multiplication, division operation | $\#i = \#i \times \#j \div \#k$                     |
| G65    | H27    | Composite square root                        | $\#i = \sqrt{\#j^2 + \#k^2}$                        |
| G65    | H31    | Sine                                         | $\#i = \#j \times \text{SIN}(\#k)$                  |
| G65    | H32    | Cosine                                       | $\#i = \#j \times \text{COS}(\#k)$                  |
| G65    | H33    | Tangent                                      | $\#i = \#j \times \text{TAN}(\#k)$                  |
| G65    | H34    | Arc tangent                                  | $\#i = \text{ATAN}(\#j / \#k)$                      |
| G65    | H80    | Unconditional transfer                       | Turn to N                                           |
| G65    | H81    | Conditional transfer 1                       | IF $\#j = \#k$ , GOTON                              |
| G65    | H82    | Conditional transfer 2                       | IF $\#j \neq \#k$ , GOTON                           |
| G65    | H83    | Conditional transfer 3                       | IF $\#j > \#k$ , GOTON                              |
| G65    | H84    | Conditional transfer 4                       | IF $\#j < \#k$ , GOTON                              |
| G65    | H85    | Conditional transfer 5                       | IF $\#j \geq \#k$ , GOTON                           |
| G65    | H86    | Conditional transfer 6                       | IF $\#j \leq \#k$ , GOTON                           |
| G65    | H99    | P/S alarm                                    | 400+N P/S alarms                                    |

## 1. Operation command

- 1) Variable assignment :  $\#I = \#J$

### G65 H01 P#I Q#J

(Example) G65 H01 P#201 Q1005; ( $\#201 = 1005$ )

G65 H01 P#201 Q#210; ( $\#201 = \#210$ )

G65 H01 P#201 Q-#202; ( $\#201 = -\#202$ )

- 2) Addition operation:  $\#I = \#J + \#K$

### G65 H02 P#I Q#J R#K

(Example) G65 H02 P#201 Q#202 R15; ( $\#201 = \#202 + 15$ )

- 3) Subtraction operation:  $\#I = \#J - \#K$

### G65 H03 P#I Q#J R#K;

(Example) G65 H03 P#201 Q#202 R#203; ( $\#201 = \#202 - \#203$ )

- 4) Multiplication :  $\#I = \#J \times \#K$

### G65 H04 P#I Q#J R#K;

(Example) G65 H04 P#201 Q#202 R#203; ( $\#201 = \#202 \times \#203$ )

5) Division operation:  $\# I = \# J \div \# K$

**G65 H05 P#I Q#J R#K**

(Example) G65 H05 P#201 Q#202 R#203; ( $\#201 = \#202 \div \#203$ )

6) Logic addition (OR):  $\# I = \# J \text{ OR } \# K$

**G65 H11 P#I Q#J R#K;**

(Example) G65 H11 P#201 Q#202 R#203; ( $\#201 = \#202 \text{ OR } \#203$ )

7) Logic multiplication (AND):  $\# I = \# J \text{ AND } \# K$

**G65 H12 P#I Q#J R#K;**

(Example) G65 H12 P# 201 Q#202 R#203; ( $\#201 = \#202 \text{ AND } \#203$ )

8) XOR:  $\# I = \# J \text{ XOR } \# K$

**G65 H13 P#I Q#J R#K**

(Example) G65 H13 P#201 Q#202 R#203; ( $\#201 = \#202 \text{ XOR } \#203$ )

9) Square root:  $\# I = \sqrt{\# J}$

**G65 H21 P#I Q#J ;**

(Example) G65 H21 P#201 Q#202 ; ( $\#201 = \sqrt{\#202}$ )

10) Absolute value:  $\# I = |\# J|$

**G65 H22 P#I Q#J ;**

(Example) G65 H22 P#201 Q#202 ; ( $\#201 = |\#202|$ )

11) Remainder :  $\# I = \# J - \text{TRUNC}(\#J/\#K) \times \# K$ , TRUNC: trunc

**G65 H23 P#I Q#J R#K**

(Example) G65 H23 P#201 Q#202 R#203; ( $\#201 = \#202 - \text{TRUNC}(\#202/\#203) \times \#203$ )

12) Decimal be converted into binary system:  $\# I = \text{BIN}(\# J)$

**G65 H24 P#I Q#J ;**

(Example) G65 H24 P#201 Q#202 ; ( $\#201 = \text{BIN}(\#202)$ )

13) Binary be converted into decimal system:  $\# I = \text{BCD}(\# J)$

**G65 H25 P#I Q#J ;**

(Example) G65 H25 P#201 Q#202 ; ( $\#201 = \text{BCD}(\#202)$ )

14) Composite multiplication, division operation:  $\# I = (\# I \times \# J) \div \# K$

**G65 H26 P#I Q#J R# k;**

(Example) G65 H26 P#201 Q#202 R#203; ( $\#201 = (\# 201 \times \# 202) \div \# 203$ )

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

**G65 H27 P#I Q#J R# k;**

(Example) G65 H27 P#201 Q#202 R#203; ( $\#201 = \sqrt{\#202^2 + \#203^2}$ )

16) Sine:  $\# I = \# J \cdot \text{SIN}(\# K)$  (Unit: deg)

**G65 H31 P#I Q#J R#K;**

(Example) G65 H31 P#201 Q#202 R#203; ( $\#201 = \#202 \cdot \text{SIN}(\#203)$ )

17) Cosine:  $\# I = \# J \cdot \text{COS}(\# K)$  (Unit: deg)

**G65 H32 P#I Q#J R# k;**

(Example) G65 H32 P#201 Q#202 R#203; ( $\#201 = \#202 \cdot \text{COS}(\#203)$ )

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

**G65 H33 P#I Q#J R# K;**

(Example) G65 H33 P#201 Q#202 R#203; (#201 = #202 • TAM (#203))

19) Arc tangent: # I = ATAN (# J / # K) (Unit: deg)

**G65 H34 P#I Q#J R# k;**

(Example) G65 H34 P#201 Q#202 R#203; (#201 = ATAN (#202/#203))

## 2. Transfer command

a) Unconditional transfer

**G65 H80 Pn; n: serial number**

(Example) G65 H80 P120; (transfer to N120 block)

b) Conditional transfer 1 #J.EQ.# K ( = )

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

(Example) G65 H81 P1000 Q#201 R#202;

# 201 = #202: transfer to N1000 block; #201 ≠ #202: the system orderly executes the blocks.

c) Conditional transfer 2 #J.NE.# K ( ≠ )

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

(Example) G65 H82 P1000 Q#201 R#202;

# 201 ≠ #202: transfer to N1000 block; #201 = #202: the system orderly executes blocks.

d) Conditional transfer 3 #J.GT.# K ( > )

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

(Example) G65 H83 P1000 Q#201 R#202;

# 201 > #202: transfer to N1000 block; #201 ≤ #202: the system orderly executes blocks.

e) Conditional transfer 4 #J.LT.# K ( < )

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

(Example) G65 H84 P1000 Q#201 R#202;

# 201 < #202: transfer to N1000 block; #201 ≥ #202: the system orderly executes blocks..

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

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

(Example) G65 H85 P1000 Q#201 R#202;

# 201 ≥ #202: transfer to N1000 block; #201 < #202: the system orderly executes blocks..

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

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

(Example) G65 H86 P1000 Q#201 R#202;

# 201 ≤ #202: transfer to N1000 block; #201 > #202: the system orderly executes blocks.

h) P/S alarm

**G65 H99 Pi i: alarm number +400**

(Example) G65 H99 P15

P/S 415 alarm.

**Note 1:** When the serial number of transferred address is specified to the positive, the system firstly searches in

positive direction and then in reverse direction; when it is specified to the negative, it firstly does in reverse direction, and in positive direction.

**Note 2:** Variable can specifies the serial number.

**G65 H81 P#200 Q#201 R#202;**

When the condition satisfies, the block transfers the block which serial number is specified by #200.

## 7.4 Notes for User Macro Program Body

### 1. Key board input

Press # following the address G, X, Z, U, W, R, I, K, F, H, M, S, T, P, Q, and # is input.

### 2. The macro program cannot be commanded in MDI mode, otherwise, the system alarms.

### 3. Operation, transfer command H.P.Q.R can be input anywhere in G65 block.

H02 G65 P#200 Q#201 R#202; ...correct

N100 G65 H01 P#200 Q10; ...correct

### 4. Single block

Generally, the operation, transfer blocks are executed, the single block switch is ON, the system does not stop. NO:13 #0 SBKM can make the single block stop. Such is applied to macro program debugging.

### 5. A variable value should be -2P32P~+2P32P-1, but the system only displays the range -9999.999~9999.999. The exceeded range is displayed to "\*\*\*\*\*".

### 6. A subprogram can execute quadruple nest.

### 7. The variable value rounds numbers, and decimal part of operation result must be rounded.

### 8. Execution time of operation, transfer command is different, and the general average time is 10ms.

## 7.5 User Macro Program Example

### 1) Custom code M81 (automatic feed)

#### Main program:

O0001;

N10 G00 X100 Z100; (Positioning to safety position)

N20 G00 U50 F100; (Rapid positioning)

N30 G01 U0.8 (Feed)

N40 M81; (Call 09081 subprogram)

N50 G0 X100 Z100; (Feed end, coordinates returning to zero)

N60 M99; (Repetitive executeion)

#### Subprogram (custom M81):

O9081;

N10 G65 H01 P#1104 Q1; (U04=1: output feed signal indication)

G65 H82 P20 Q#1004 R1; (Whether X14=1, X14=0: N20 program is executed)

G65 H01 P#1100 Q0; (Cancel feed signal indication U04=0)

M99 P50; (Transfer to main program N50)

N20 M99 P30; (Transfer to main program N30)

## Chapter Eight Tool Compensation C Function

The actual tool nose is not a point but an arc. Influenced by the tool nose arc, an error occurs between the actual machined result and the workpiece program, and the tool compensation C function can realize the tool radius compensation to clear the above error. The corresponding parameter must be set to activate the function when the tool compensation C function is executed: NO:2#3 is set to 1.

|   |   |   |  |  |  |  |      |  |  |  |  |
|---|---|---|--|--|--|--|------|--|--|--|--|
| 0 | 0 | 2 |  |  |  |  | CCMP |  |  |  |  |
|---|---|---|--|--|--|--|------|--|--|--|--|

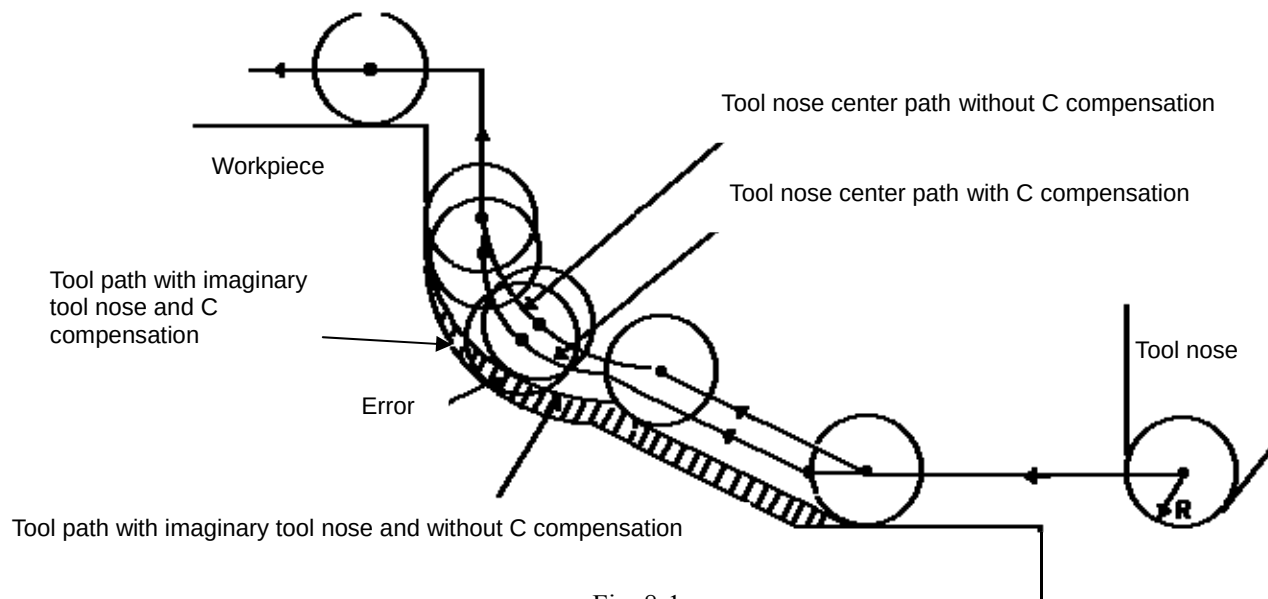


Fig. 8-1

### 8.1 Basic Concept of Tool Compensation C Function

#### 8.1.1 Imaginary Tool Nose

The tool nose A point described in Fig. 8-1-1-1 is imaginary, and does not exist actually, and is called imaginary tool nose. Generally, because it is difficult to set the tool nose radius center on the initial position, and it is easy to set the imaginary tool nose on the initial position, which is shown below. Do not concern the tool nose radius when the imaginary tool nose is executed in programming.

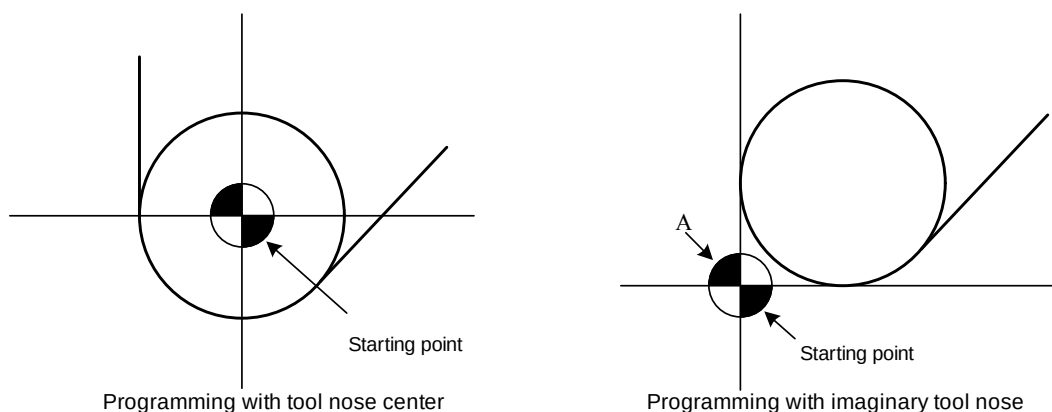


Fig. 8-1-1-1 tool nose radius center and imaginary tool nose

**Note:** For the machine with machine zero, a standard point such as tool post center is taken as a starting point. The distance from the standard point to the tool nose radius center or imaginary tool nose is the tool offset value.

The distance between the standard point and the tool nose radius center is offset value. For setting the tool offset value, it is easier to measure the distanced between the standard point to imaginary tool nose than the standard point to the tool nose radius center, generally, the distance between the standard point and imaginary tool nose is set to the tool offset value.

When the tool post center is taken as a starting point, the tool offset value is shown below:

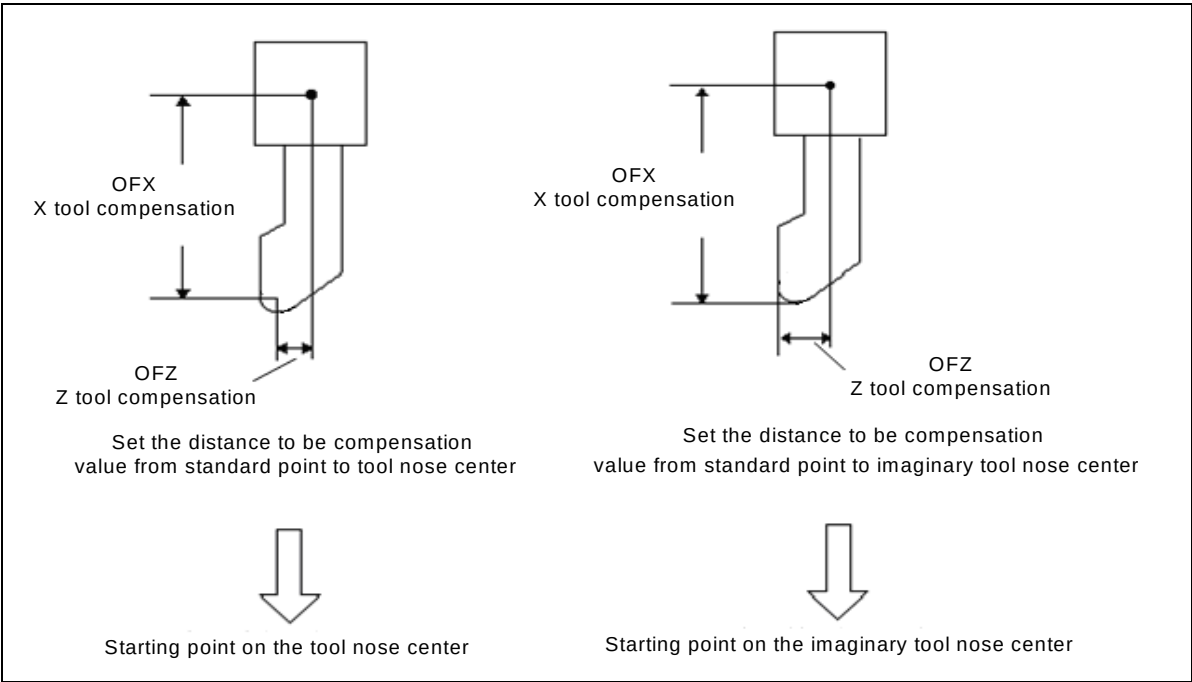


Fig. 8-1-1-2 tool offset value setting when tool post center as standard point

Fig. 8-1-1-3 and Fig. 8-1-1-4 are tool paths of tool nose center programming and imaginary tool nose programming. The left figure is radius compensation without tool nose radius, the right is one with tool nose radius compensation.

The imaginary tool nose path is the same as the programmed path without the tool nose radius compensation

use tool nose radius compensation used to as the precise cutting

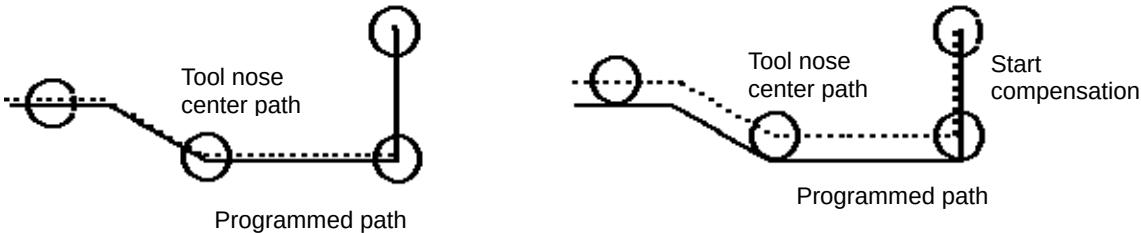


Fig. 8-1-1-3 tool path in tool nose center programming

The imaginary tool nose path is the same as the programmed path without the tool nose radius compensation

use tool nose radius compensation used to as the precise cutting

programmed path without the tool

precise cutting

nose radius compensation

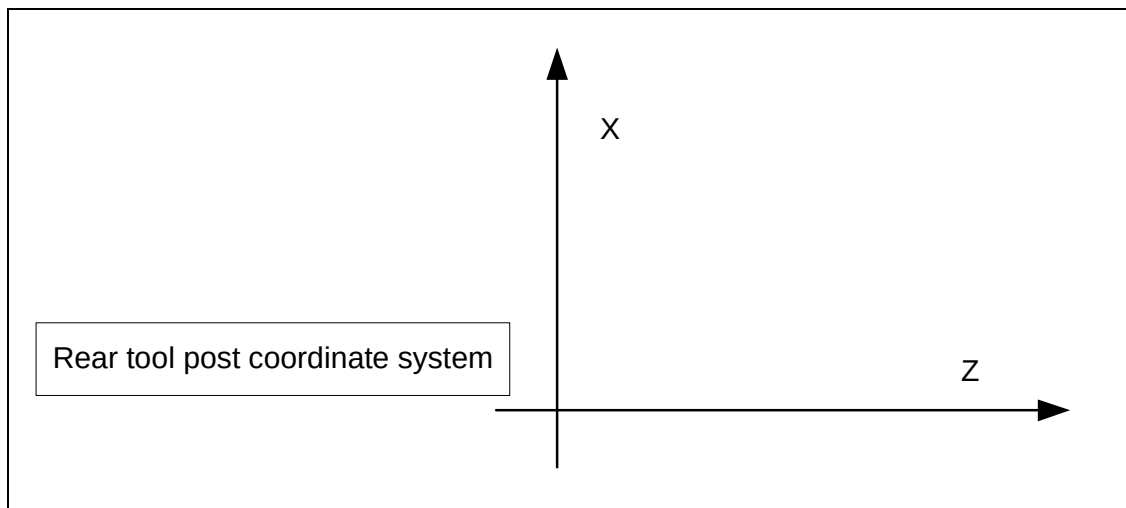


Fig. 8-1-1-4 Tool path in imaginary tool nose programming

### 8.1.2 Imaginary Tool Nose Direction

In actual machining, there are different position relations between the tool and workpiece. The imaginary tool nose direction is determined by the tool direction when cutting.

The imaginary tool nose number defines the position relations between the imaginary tool point and tool nose arc center. The imaginary tool nose number is divided into 10 kinds (0~9), meaning position relations in 9 directions. The tool nose directions are different in different coordinate system (rear tool post coordinate system and front tool post coordinate system) even if they are the same tool nose direction numbers as the following figures. In figures, it represents relationships between tool nose and starting point, and end point of arrowhead is the imaginary tool nose.



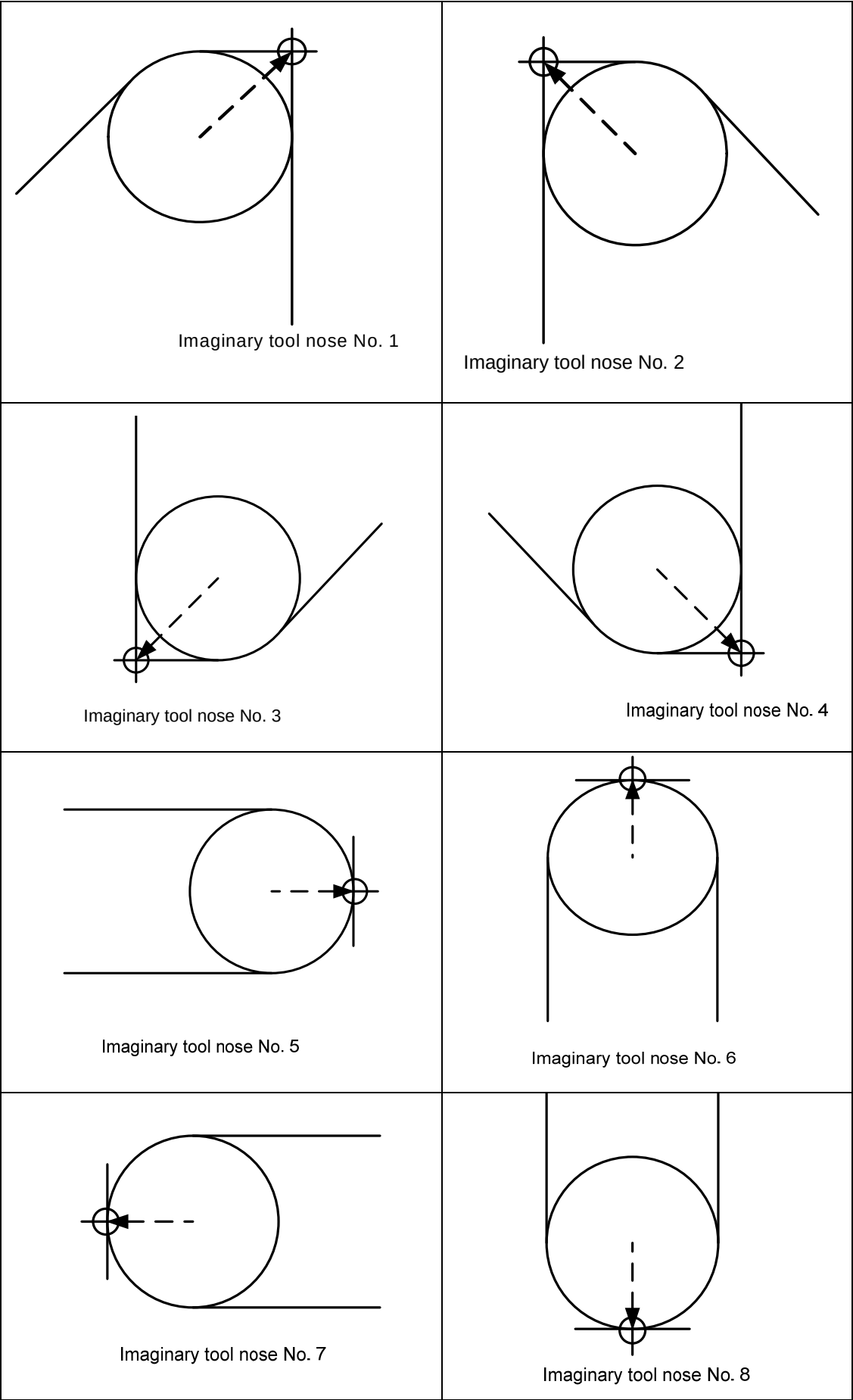
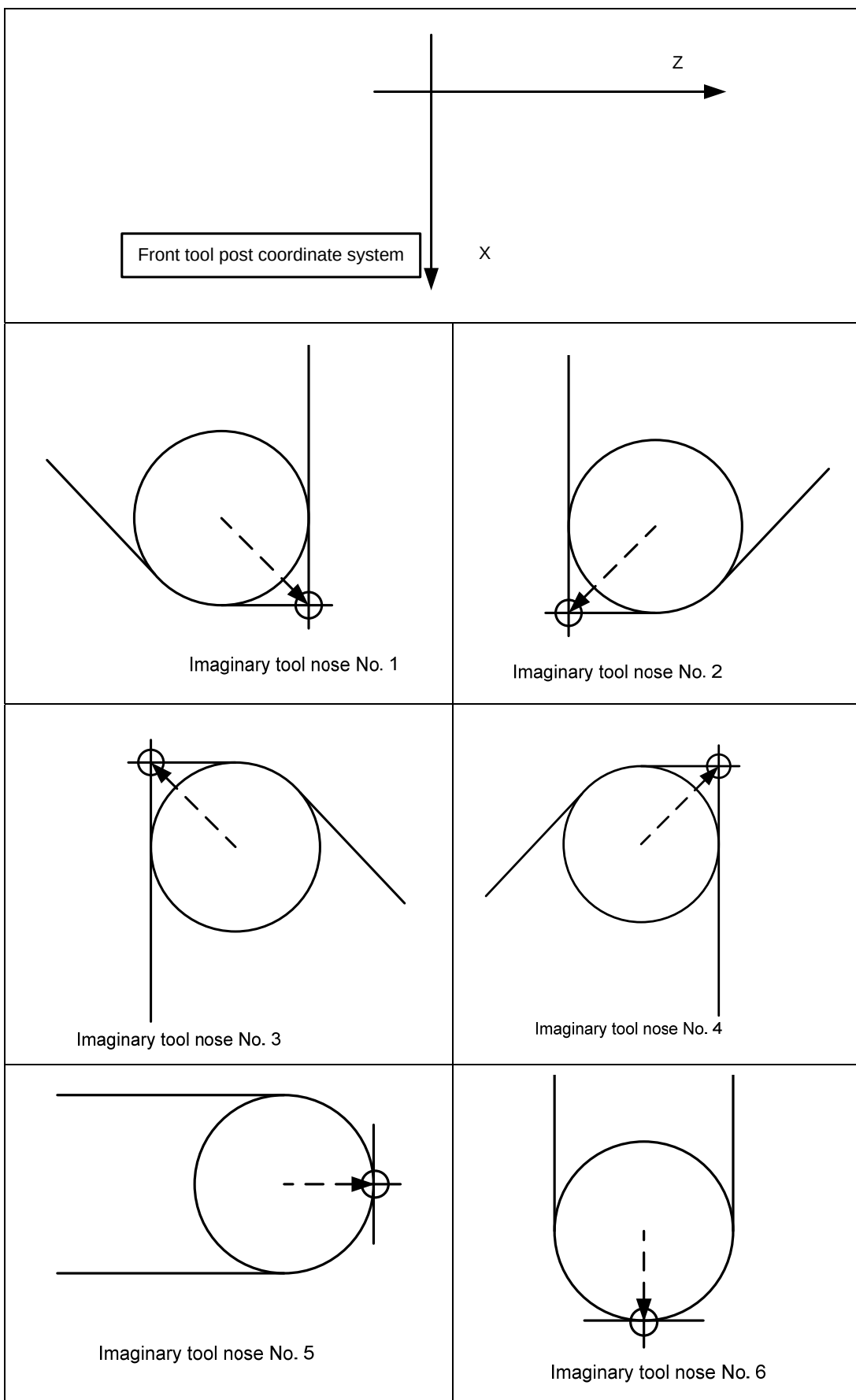


Fig. 8-1-2-1 Imaginary tool number in rear tool post coordinate system



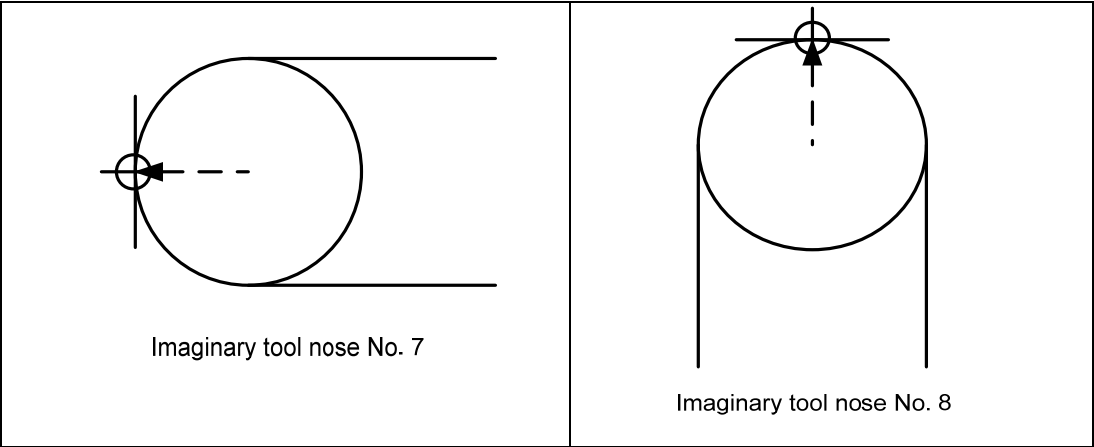


Fig. 8-1-2-2 imaginary tool nose number in front tool post coordinate system

When the tool nose center and the starting point are consistent, the tool nose number is set to 0 or 9. The corresponding tool compensation number uses the address T to set the imaginary tool nose number.

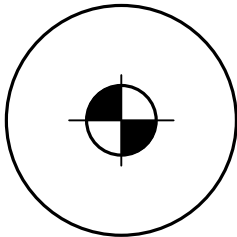


Fig. 8-1-2-3 tool nose center and starting point be consistent

8.1.3 Compensation Value Setting

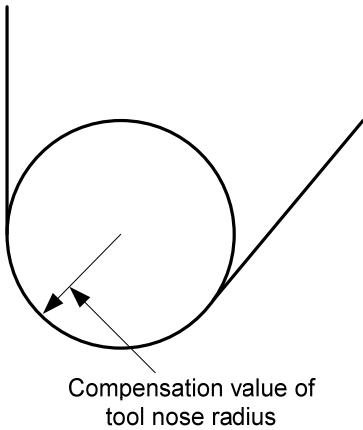


Fig. 8-1-3-1 compensation value of tool nose radius

Set the following compensation values before executing the tool nose radius compensation: X, Z, R, T. X, Z are tool offset values from the tool post center to the tool nose in X, Z direction; R is the radius compensation value of imaginary tool nose; T is the imaginary tool nose number. One value corresponds to a tool compensation number, which is set in tool compensation window. See Modify and Set Tool Compensation Value.

The concrete is shown in Table 8-1-3-1:

Table 8-1-3-1 tool nose radius compensation value display window

| NNo. | X     | Z     | R     | T  |
|------|-------|-------|-------|----|
| 001  | 0.020 | 0.030 | 0.020 | 2  |
| 002  | 0.060 | 0.060 | 0.016 | 3  |
| ..   | ..    | ..    | ..    | .. |
| ..   | ..    | ..    | ..    | .. |
| ..   | ..    | ..    | ..    | .. |
| 015  | 0.030 | 0.026 | 0.18  | 9  |
| 064  | 0.050 | 0.038 | 0.20  | 1  |

#### 8.1.4 Relative Position between Tool and Workpiece

The relative position between the tool and workpiece must be specified when the tool radius compensation is executed. In rear tool post coordinate system, the tool center path is the right of the programmed path (part path) advancing direction, which is called right tool compensation, using G42; the tool center path is the left of the programmed path (part path) advancing direction, which is called left tool compensation, using G41. In front tool post system, they are reverse. The relative position is shown in Table 8-1-4-1 when G40, G41, G42 is commanded.

Table 8-1-4-1

| Command | Function Explanation                                                                                                                             | Remark |
|---------|--------------------------------------------------------------------------------------------------------------------------------------------------|--------|
| G40     | Cancel tool nose radius compensation                                                                                                             |        |
| G41     | Tool nose radius left compensation in rear tool post coordinate system, tool nose radius right compensation in front tool post coordinate system |        |
| G42     | Tool nose radius right compensation in rear tool post coordinate system, tool nose radius left compensation in front tool post coordinate system |        |

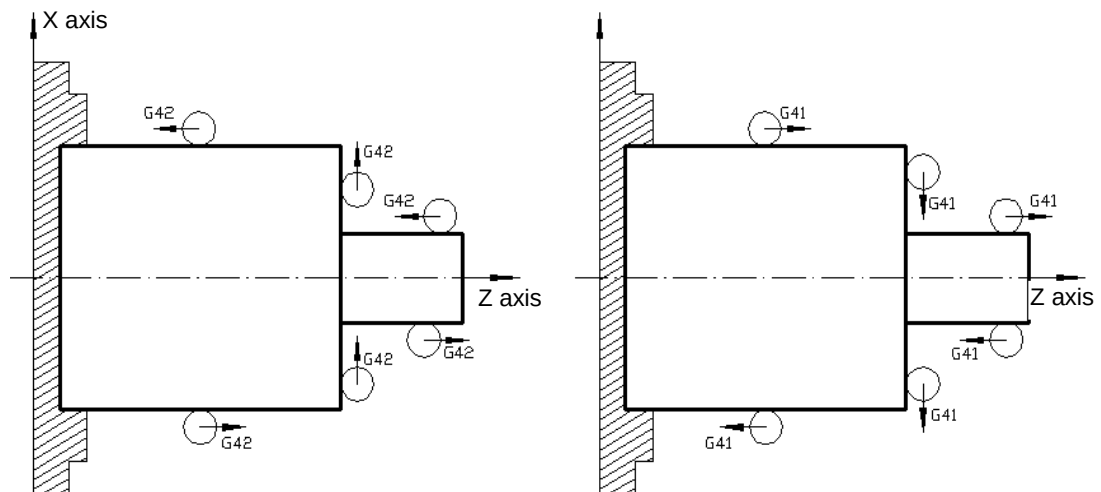
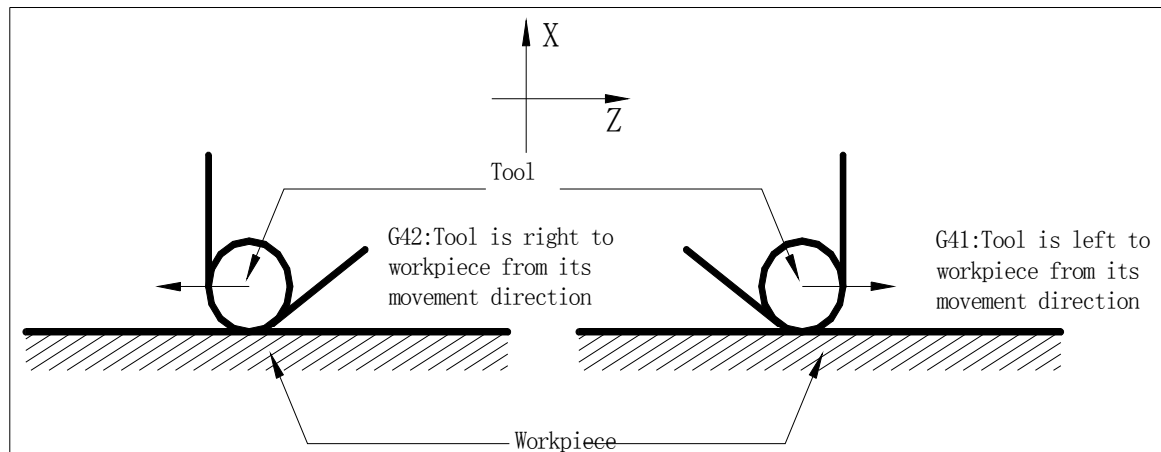


Fig. 8-14-1 tool nose radius compensation in rear tool post coordinate system

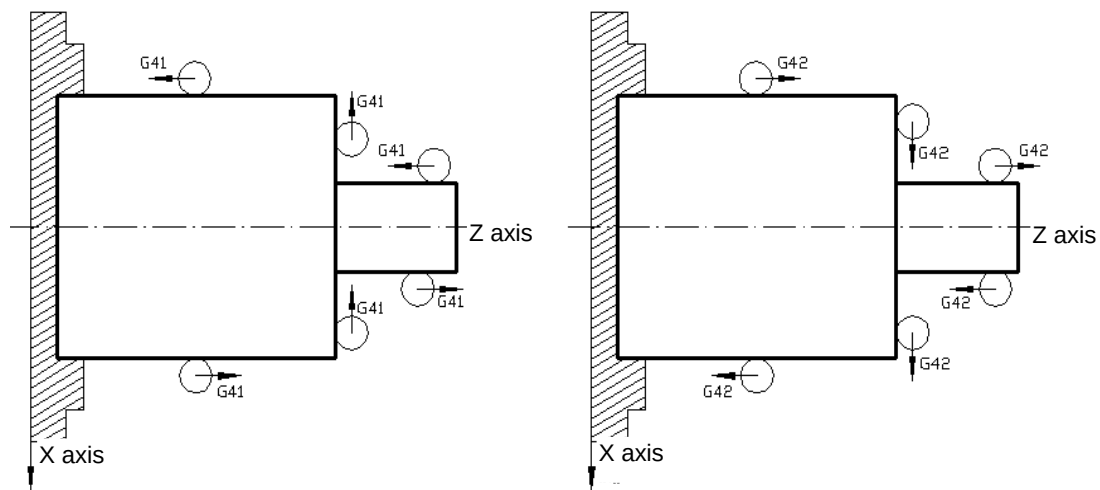
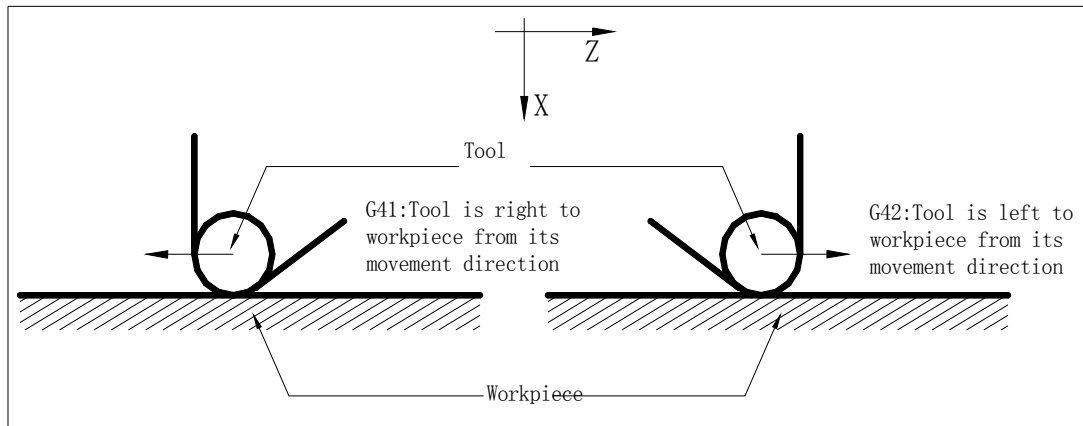


Fig. 8-1-4-2 tool nose radius compensation in front tool post coordinate system

### 8.1.5 Inside and outside

When the tool nose radius compensation is executed, the corners of previous and current programmed path are different, and their tool nose compensation paths are so. The intersection angle of the block intersection of two movement blocks at the side of workpiece is more than or equal to  $180^\circ$ , which is called inside, the intersection angle is  $0 \sim 180^\circ$ , which is called outside.

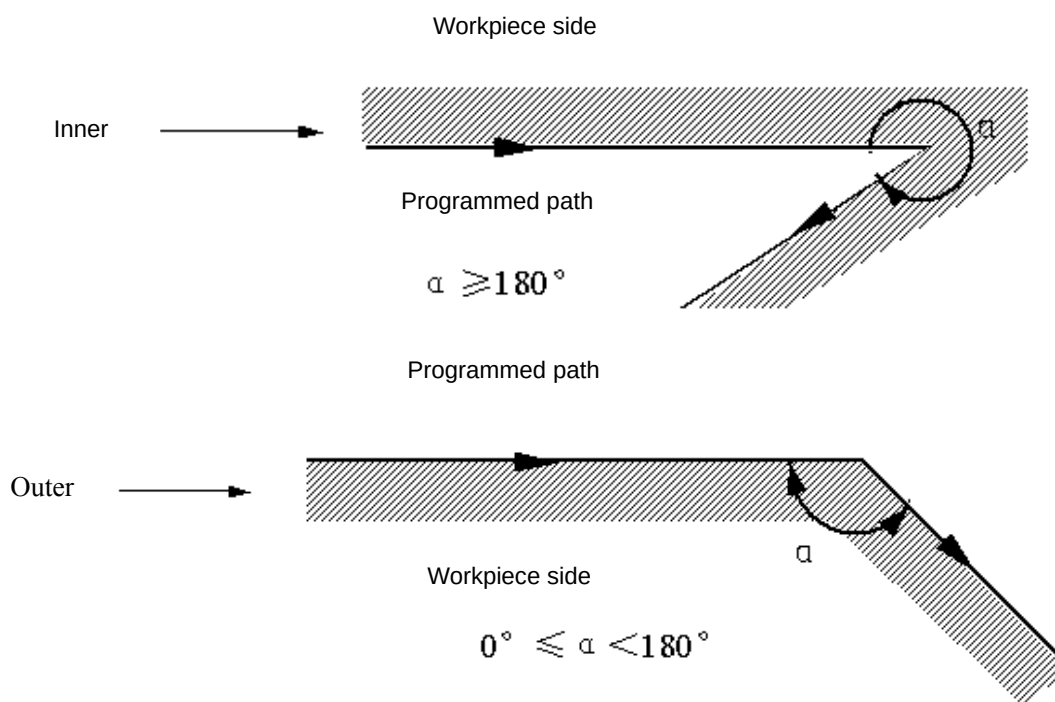


Fig. 8-1-5-1 Inner and outside

### 8.1.6 Command Format of G41, G42, G40

**Command format:**

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

**Note 1:** G40, G41, G42 are modal.

**Note 2:** G02/G03 following G41/G42 can be used after the tool compensation is established normally.

## 8.2 Concrete Tool Compensation

### 8.2.1 Concrete Path Decomposition of Tool Nose Radius Compensation

The tool radius compensation steps are divided into three: tool compensation establishment, execution and cancel.

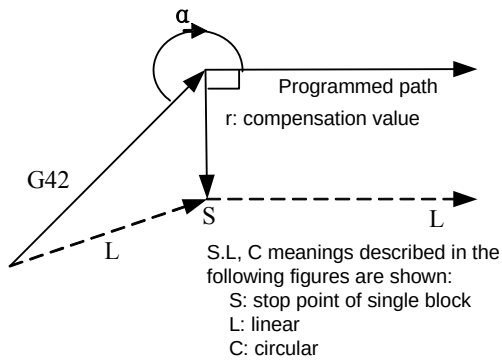
#### 1. Tool compensation establishment

The execution from offset cancel to offset mode is called tool compensation establishment.

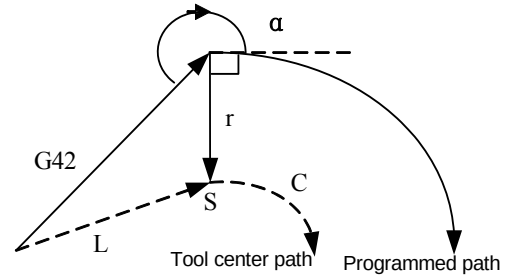
The concrete tool compensation establishment is shown in Fig. 8-2-1-1:

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

(i) Linear—linear

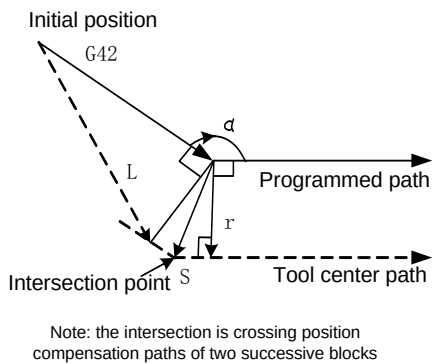


(ii) Linear—circular

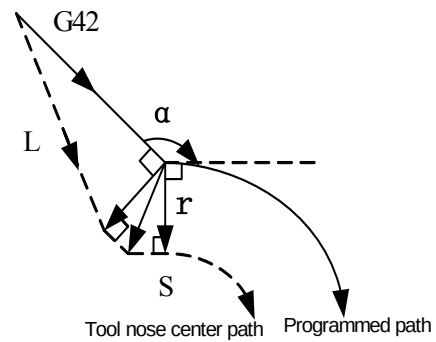


b) Tool traversing outside along corner ( $180^\circ > \alpha \geq 90^\circ$ )

(i) Linear-linear

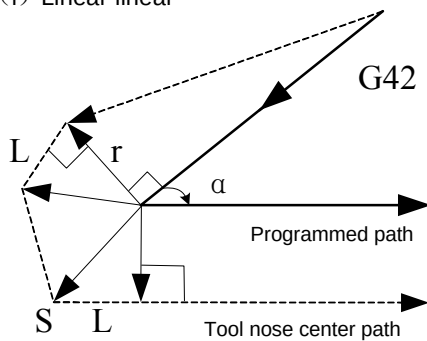


(ii) Linear—circular

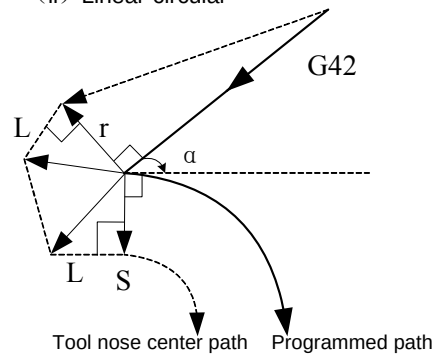


(c) Tool traversing outside along corner ( $\alpha < 90^\circ$ )

(i) Linear-linear



(ii) Linear-circular



(d) Tool traversing outside along corner, linear→linear. ( $\alpha < 1^\circ$ )

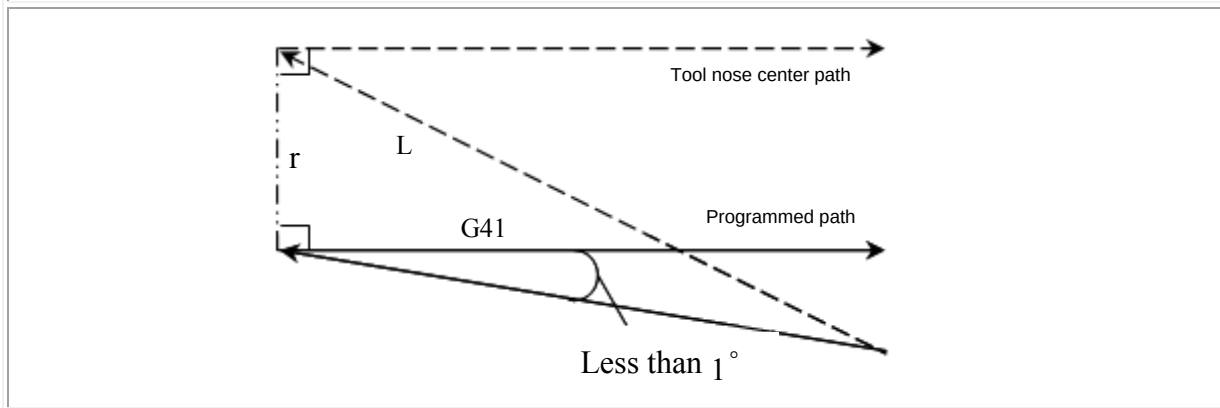


Fig. 8-2-1-1 tool compensation establishment

**Note 1:** When the tool compensation is established, the tool compensation number is not specified or is zero, the alarm #036 occurs.

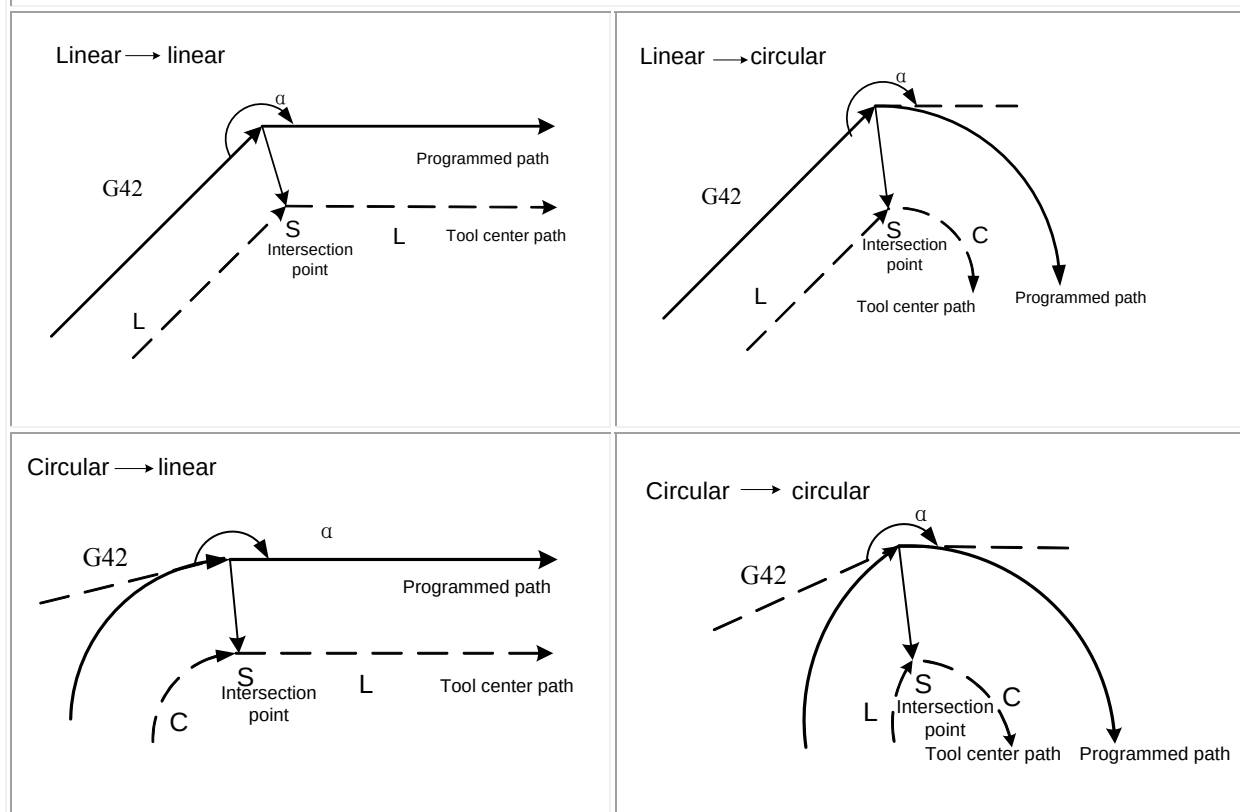
**Note 2:** When the tool compensation is established, it is executed by G0 or G1; the #034 alarm occurs when it is done by G02 or G03.

## 2. Tool compensation proceeding

The offset path after the tool compensation establishment and before the tool compensation cancel is called tool compensation proceeding.

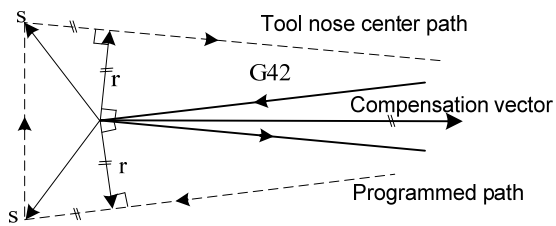
The tool compensation proceeding is shown in Fig. 8-2-1-2 and Fig. 8-2-1-3:

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



(V) Inside machining less than 1 degree and zooming out compensation vector

(i) Circular → Circular



Use the same method to the followings

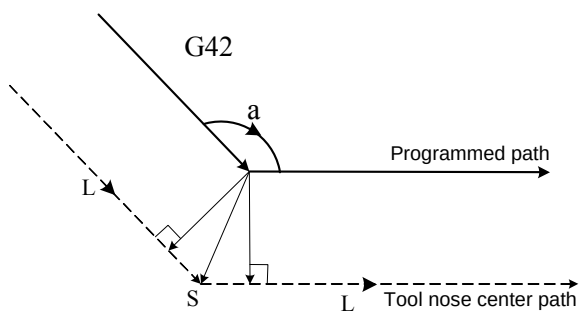
(ii) Circular → Linear

(iii) Linear → Circular

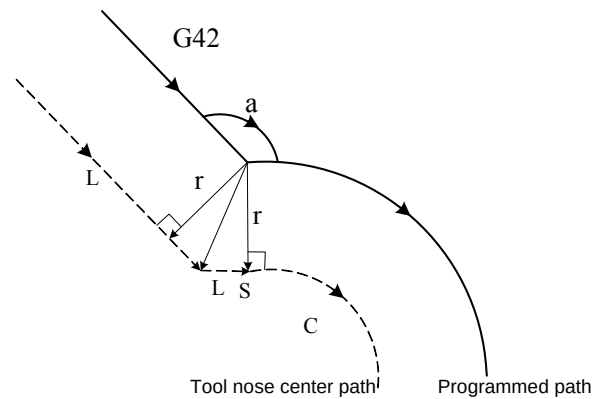
(iv) Circular → Circular

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

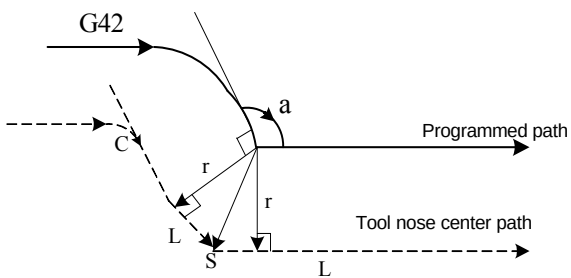
(i) Linear -- linear



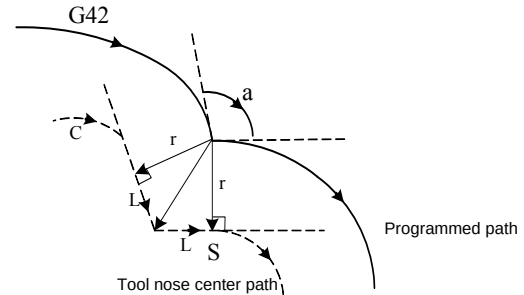
(ii) Linear -- circular



(iii) Circular -- linear



(iv) Circular -- circular



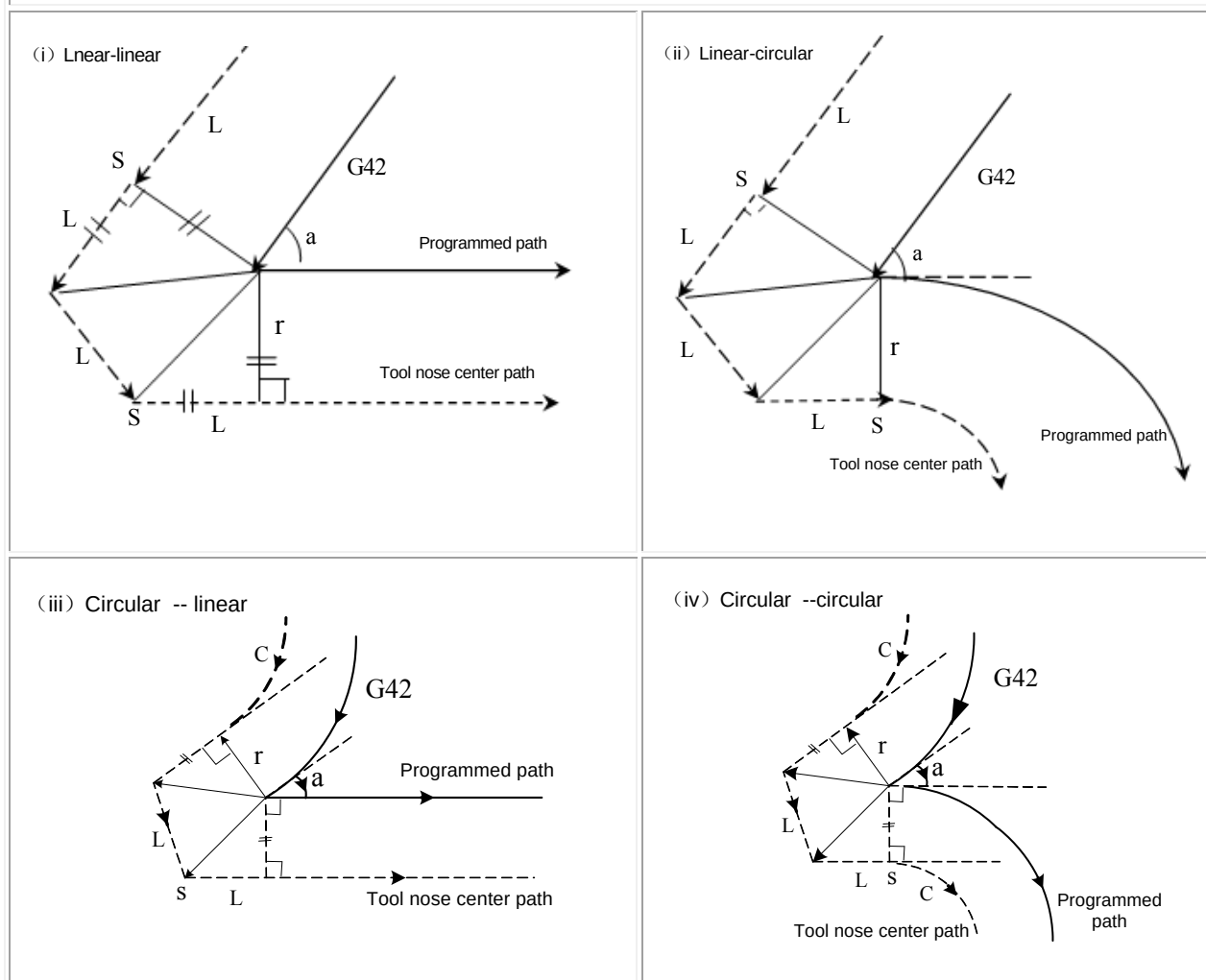
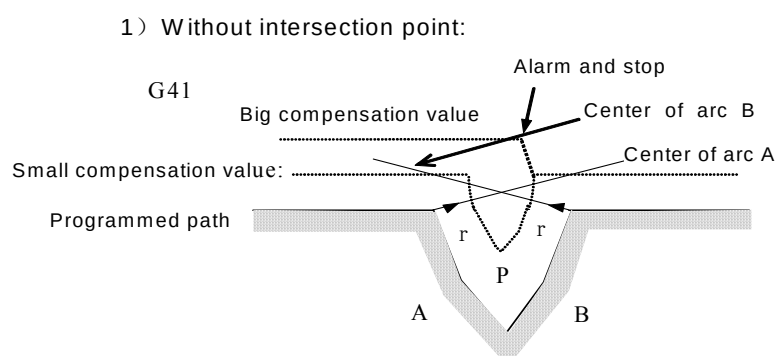
(c) Tool traversing outside along corner ( $\alpha < 90^\circ$ )


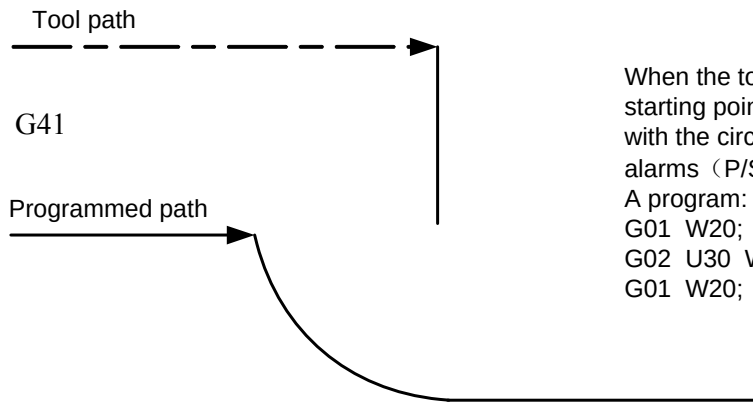
Fig. 8-2-1-2 tool compensation execution

## (d) Special



In the left figure, when the arc compensation path has an intersection point, but when the radius becomes big, the intersection point lose, at the moment, the system alarms (PIS33).

## 2) Circle center coinciding with starting point/end point



When the tool radius is excessive big, starting point or end point coincides with the circle center, the system alarms (P/S38)  
A program:  
G01 W20;  
G02 U30 W15 R15;  
G01 W20;

Fig. 8-2-1-3 tool compensation execution ②

### 3. tool compensation cancel

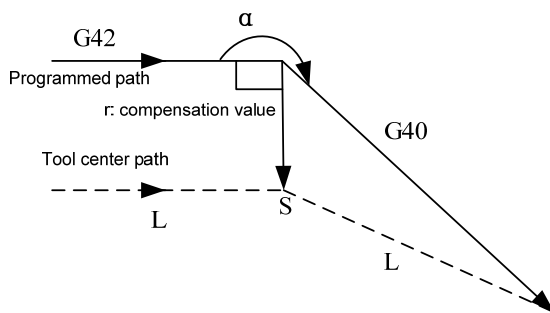
In compensation mode, when a program meets anyone of the following conditions, the system enters the compensation cancel mode, and the block operation is called tool compensation cancel.

- (a) Using G40 cancels C tool compensation, executing tool compensation cancel cannot use G02, G03, otherwise, NO. 34 alarm occurs and the tool stops.
- (b) The tool radius compensation number is specified to 0.

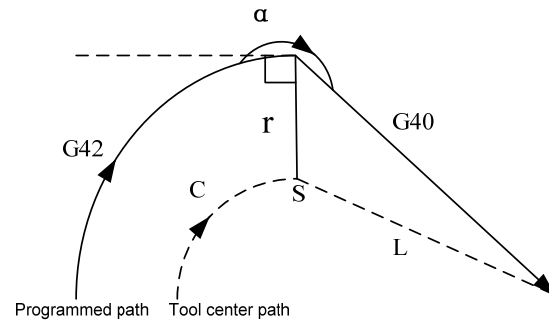
The concrete tool compensation cancel is shown below:

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

##### (i) Linear—linear



##### (ii) Circular—linear



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

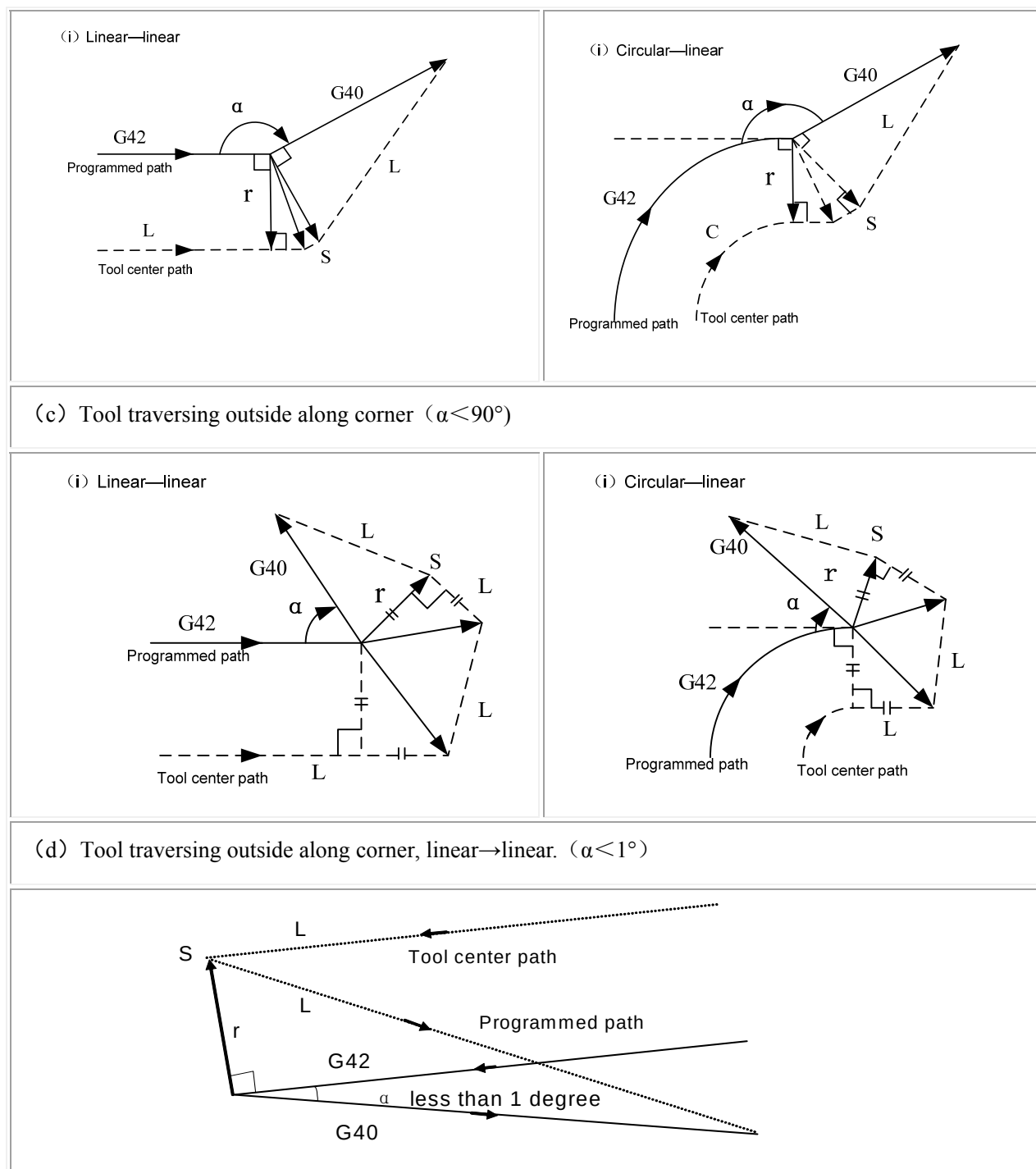


Fig. 8-2-1-4 tool compensation cancel

## 8.2.2 Compensation direction change when tool compensation proceeding

The tool compensation G code (G41 and G42) determines the compensation direction, and the compensation value sign is shown below:

Table 8-2-2-1

| Compensation value signs<br>G code | +                  | -                  |
|------------------------------------|--------------------|--------------------|
| G41                                | Left compensation  | Right compensation |
| G42                                | Right compensation | Left compensation  |

In special conditions, the compensation direction can be changed in the compensation mode, but it cannot be changed in the initial of block. The following compensation value is positive.

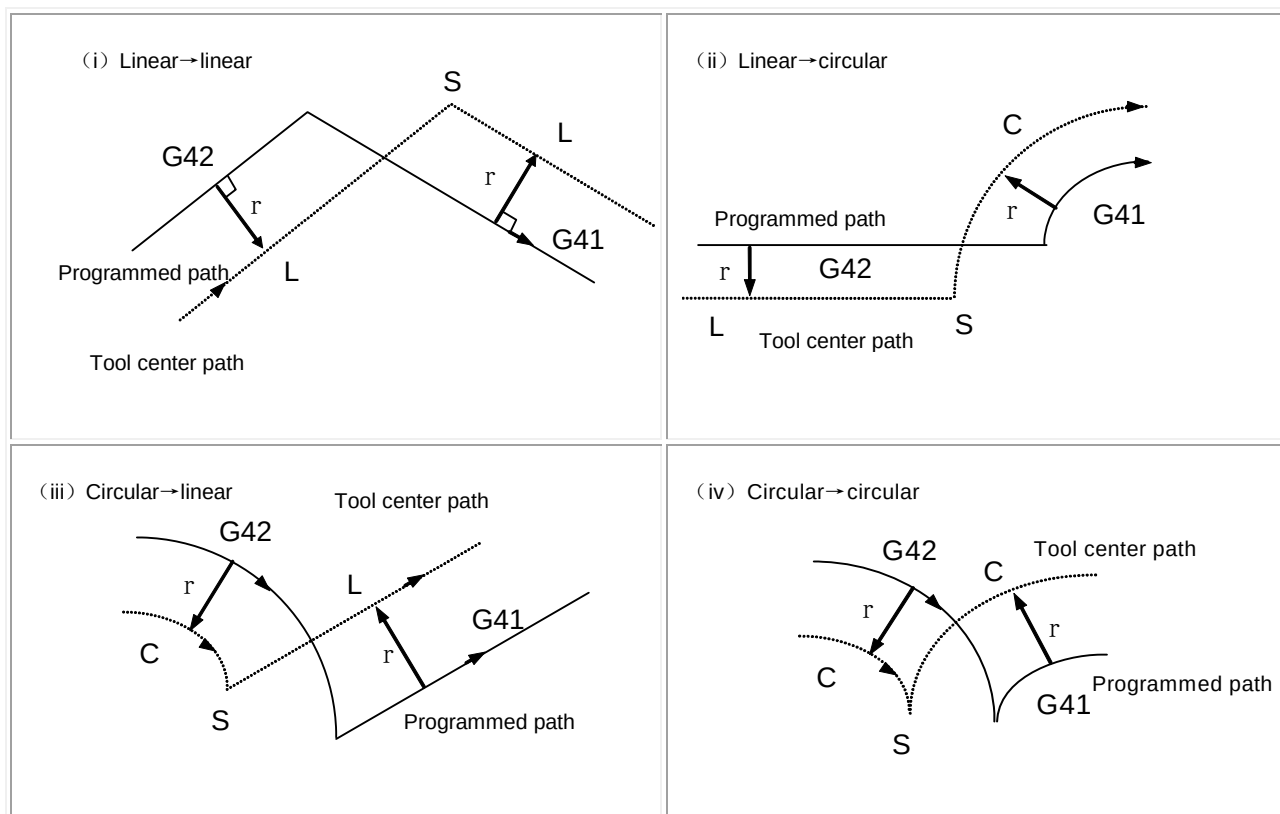


Fig. 8-2-2-1 changing compensation direction when tool compensation proceeding

When the compensation is normally executed, there is no intersection point, G41 or G42 changes compensation direction, the offset path from the block A to B is shown below:

i) Linear-----linear

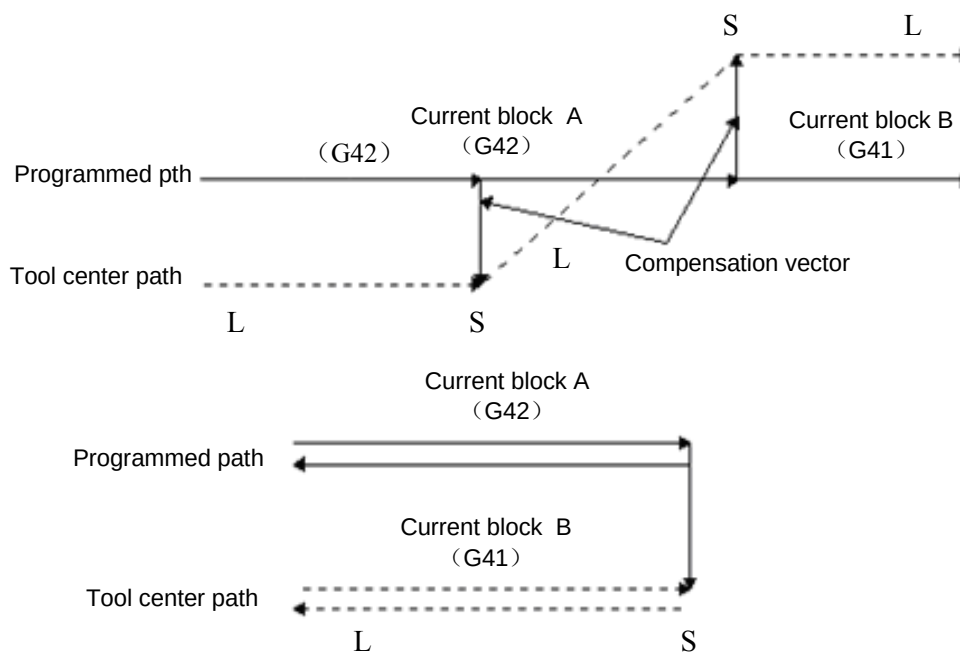


Fig. 8-2-2-2 linear — linear

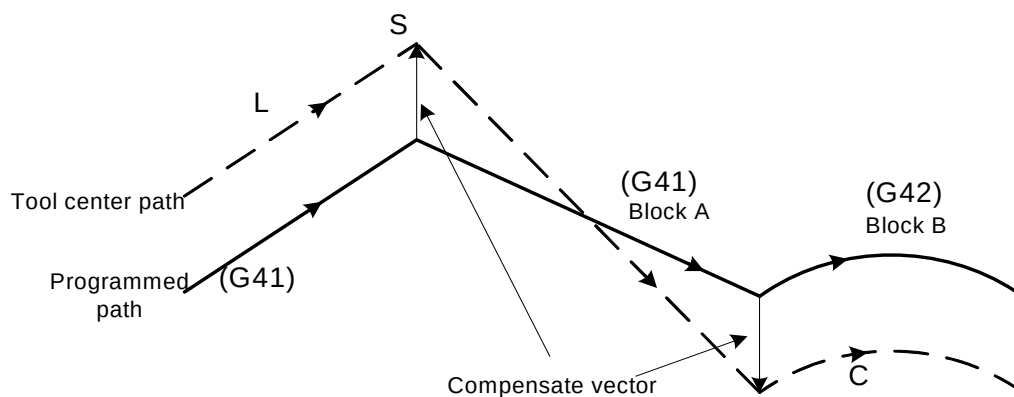
ii ) Linear-----circular

Fig. 8-2-2-3 linear—arc without intersection (change compensation direction)

**8.2.3 Temporary Tool Compensation Cancel**

In compensation mode, when the following G Codes are specified, the compensation vector is temporarily cancelled, later, it automatically recovers.

At the moment, it is different from the compensation cancel mode, the tool directly traverses from the intersection point to the command point of compensation vector cancel. When the compensation mode recovers, the tool directly traverses to the intersection point.

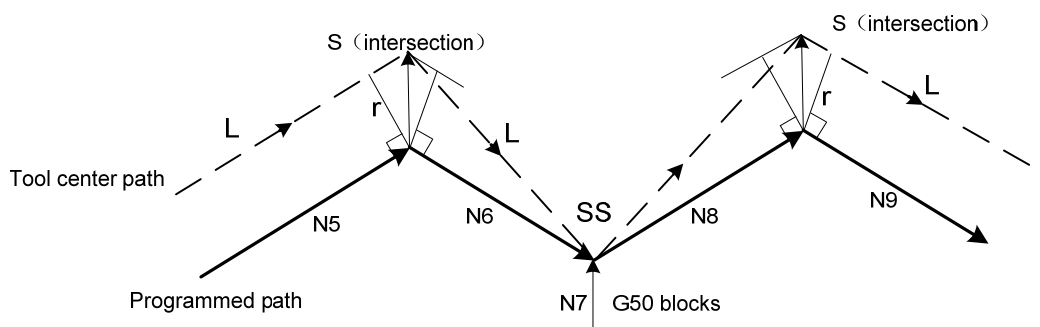
## 1. Coordinate system setting (G50)

N1 T0101;

N2 G42 G00 X0 Z0;

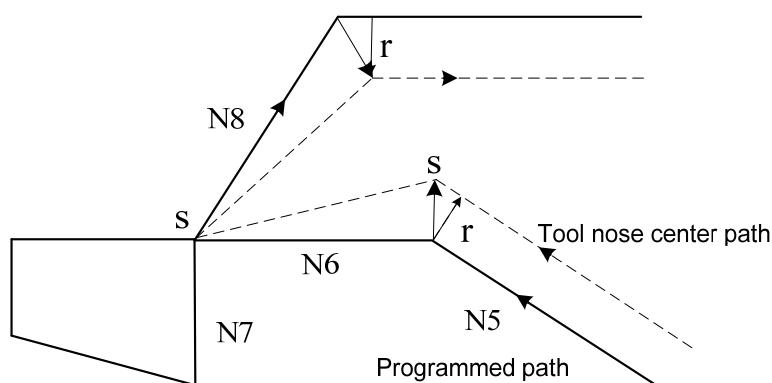
N3 G01 U-30 W30;

## Part Two Programming



Note: SS means to be the point where the tool stops twice in single block mode.

2. G90, G92, G94 are fixed cycle commands, G71~G76 are fixed cycle commands, G32/ G33/ G34 are thread machining Codes.



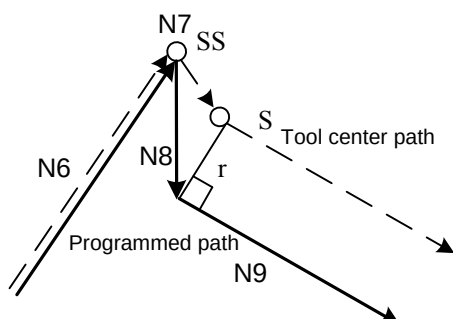
N1 T0101  
N2 G0 X100 Z100  
N3 G0 X0 Z0  
N4 G42 G90 X-20 W-50 F500 (Temporarily cancel tool compensation here)  
N5 G0 X50 Z50 (Recover tool compensation )  
N6 G0 X100 Z100  
N7 M30

**Note:** G90/G94 is temporarily cancelled and is executed only when G41/G42 and G90/G94 are in the same block, the system automatically executes G90/G94 tool compensation when they are not in the same block. See G90/G94 tool nose radius compensation in detail.

### 8.2.4 Non-Movement Command when Tool Compensation

#### 1. Non-movement command when starting compensation

The compensation vector is not executed when there are no tool traverse commands at the beginning of compensation commands.

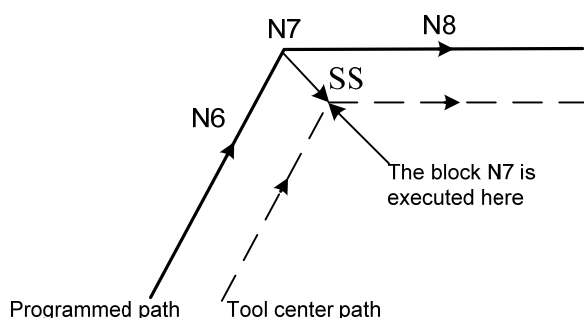


```
N1 T0101;
N2 G0 X0 Z0;
N3 G01 U-30 W20 F500;
N4 G42 U0;
N5 U30;
N6 U20 W20;
N7 G40 G0 X100 Z100;
N8 M30;
```

Fig. 8-2-4-1 non-movement command when tool compensation start

#### 2. Non movement command when executing compensation

In compensation mode, only one block without tool traversing is commanded, the vector and tool center path is the same that the block is not commanded (refer to Section 8.2.1 Tool Compensation Proceeding), the block without tool traversing is executed at the point where a single block stops.

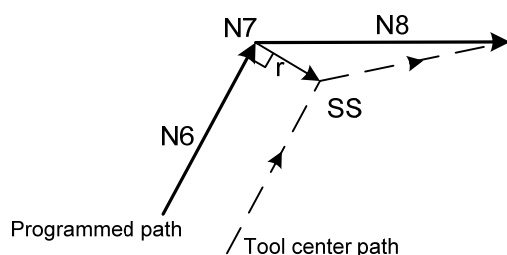


```
N3 T0101;
N4 G0 X100 Z100;
N5 G41 G01 X0 Z0;
N6 U-30 W20;
N7 G04 X5;
N8 W30;
N9 G40 G0 X100 Z100;
N10 M30;
```

Fig. 8-2-4-2 non-movement command when executing tool compensation

#### 3. Non-movement command when tool compensation cancel

When the block which is commanded with the compensation cancel has no tool traversing, its formed length is the compensation value, its direction is vertical with the vector of the previous block movement, and the vector is cancelled at the next movement command.



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

Fig. 8-2-4-3 non-movement command when tool compensation cancel

### 8.2.5 Tool Compensation Interference Check

The tool overcutting is called “interference”. The tool compensation interference check in advance the tool overcutting, namely, the interference check is executed even if the overcutting is not done.

(a) General conditions of interference

- (1) The directions of tool path and programmed path are different.(intersection angle of paths is between  $90^\circ$  and  $270^\circ$ ).
- (2) When the arc is machined, the angle between starting point and end point of tool center path is great different from that of programmed path (more than  $180^\circ$  ).

**Example ①**

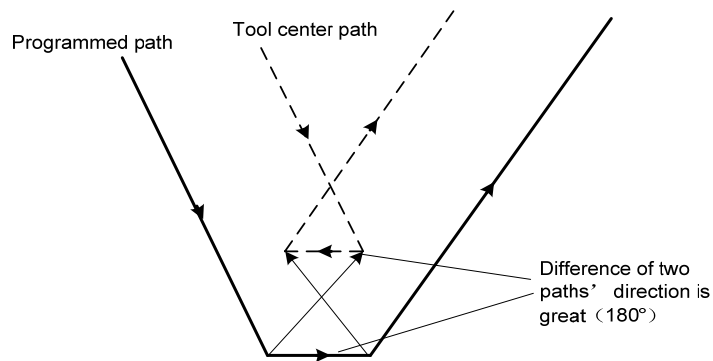


Fig. 8-2-5-1 tool compensation interference example ①

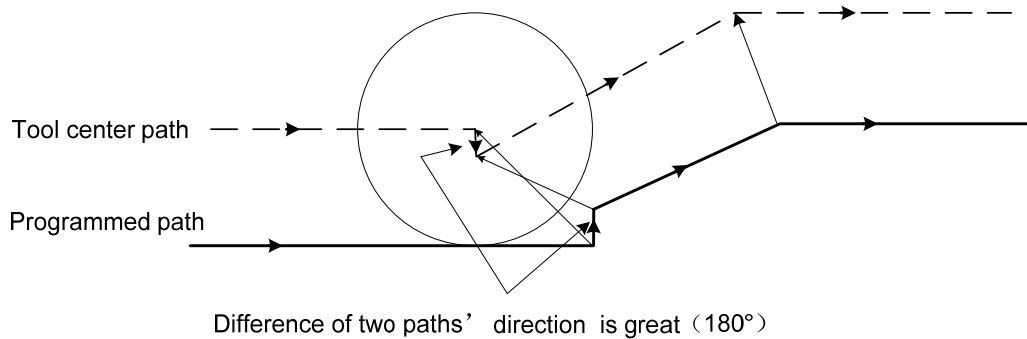


Fig. 8-2-5-2 tool compensation interference example ②

(b) Interference example

- (1) One shallow depth, less than the compensation value.

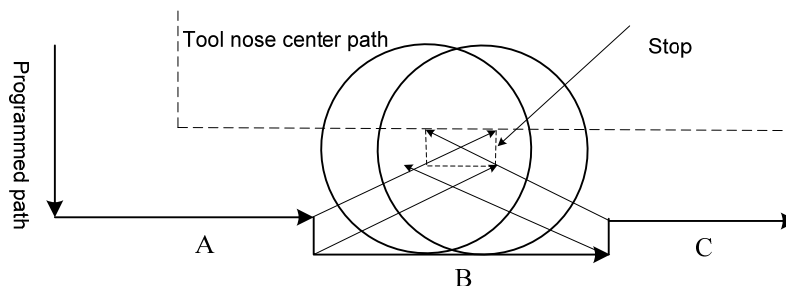


Fig. 8-2-5-3 tool compensation interference example ③

A program :

```

N1 T0101;                                (R≤10)
N2 G0 X0 Z30;
N3 G42 G01 X50 Z0 F500;
N4 U50;
N5 W20;
N6 U10;
N7 W20;
N8 U-10;
N9 W20;
N10 G40 G0 X0 Z30;
N11 M30;

```

In the above program, tool nose radius compensation value of No. 1 tool  $R \leq 10$ , the tool compensation is executed normally;  $R > 10$ : the interference alarm occurs because the block B direction is reverse to the tool radius compensation's path.

(2) Grooving depth less than compensation value

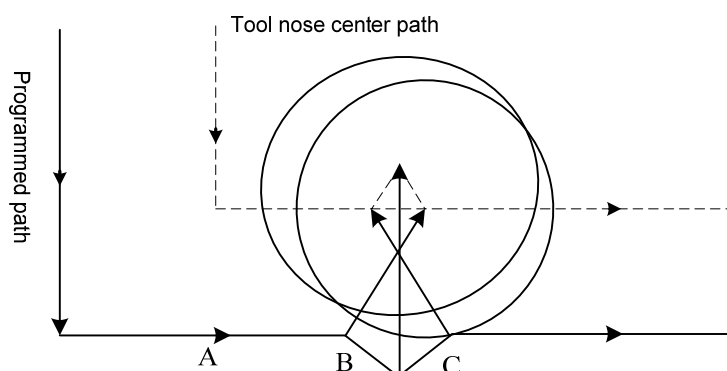


Fig. 8-2-5-4 tool compensation interference example ②

A program:

```

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

```

In the above program, tool nose radius compensation value of No. 1 tool  $R \leq 25$ , the tool compensation is executed normally;  $R > 25$ : the interference alarm occurs because the block c direction is reverse to the tool radius compensation's path.

## 8.2.6 Tool Nose Radius Compensation in G90/G94

- For all paths of cycle, the tool nose center path parallel with the programmed path.

### ① G90

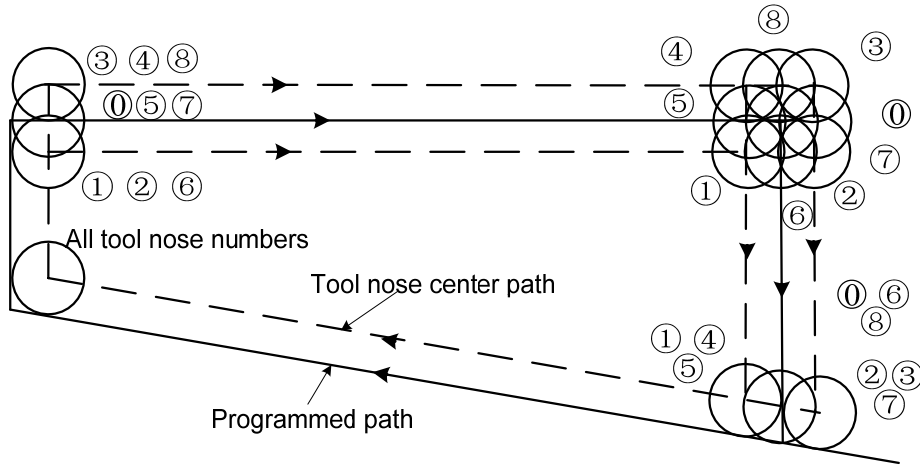


Fig. 8-2-6-1 G90 tool nose radius compensation

### ② G94

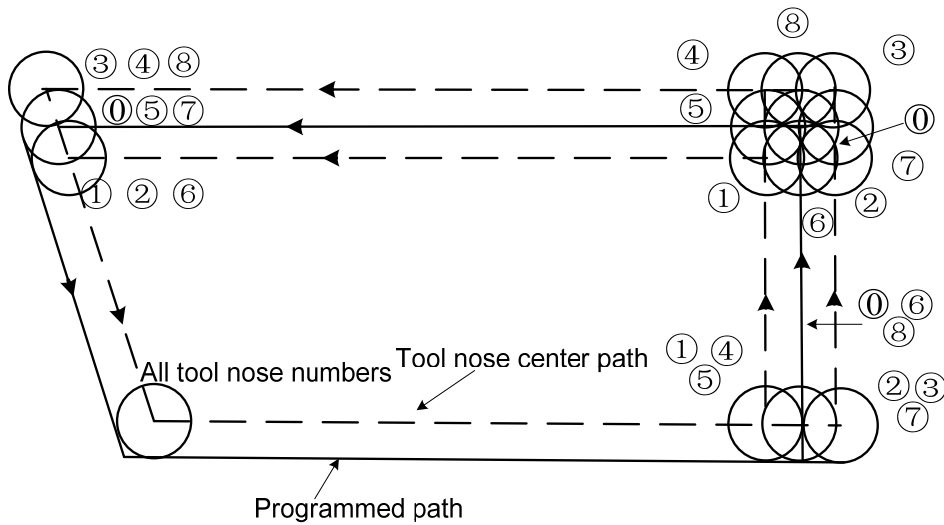


Fig. 8-2-6-2 G94 tool nose radius compensation

- In G41 or G42, the offset direction is shown below:

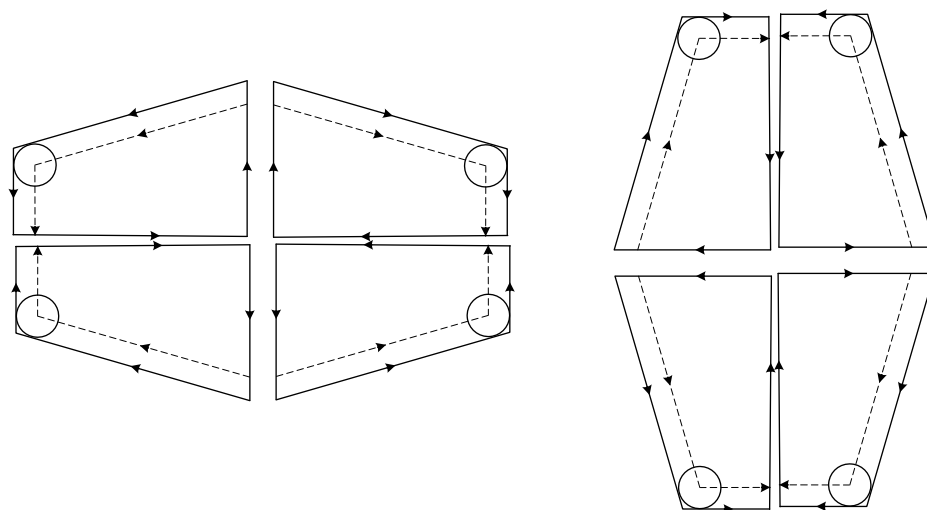


Fig. 8-2-6-3 G90/G94 tool nose radius compensation

### 8.2.7 Tool Nose Radius Compensation in G70

In G70 (finishing cycle), the tool nose radius compensation can be executed, and the tool center path automatically offsets one compensation value along finishing path. When G70 tool nose radius compensation is executed, G70 and G41/G42 are in the same block or G70 specifies G41/G42 in finishing cycle block.

## 8.3 Notes of Tool Compensation C

- The system alarms (NO.39) when 30 non-movement blocks or more are specified continuously in the course of compensation:
 

|       |                 |                        |
|-------|-----------------|------------------------|
| N1    | M05 ; .....     | M code output          |
| N2    | S21 ; .....     | S code output          |
| N3    | G04 X10 ; ..... | Dwell                  |
| ..... |                 |                        |
| N29   | G01 U0 ; .....  | Movement distance zero |
| N30   | G98 ; .....     | only G code.           |
- In MDI mode, blocks are executed but the tool nose radius compensation are not done.
- After the system is turned on or when M30 is executed, the system enters into cancel mode. The program must end in cancel mode, otherwise, the tool cannot position in the end point, stops a position which is a vector from the end point.
- Establishment or cancel of tool nose radius compensation only uses G00 or G01 instead of G02 or G03. When G02 or G03 is specified, the system alarms (NO.34).
- Before the subprogram is called (i.e. before M98 is executed), the system must enters the compensation cancel mode. After the system enters subprograms, the offset is activated, but before the system returns to main programs (i.e. M99 is executed), it must be in the compensation cancel mode, otherwise, the system alarms. (No.36).
- When compensation value (R) is negative, G4, G42 is exchanged each other. When the tool center traverses outside along the workpiece, it does inside, and vice versa. Because the compensation sign

7. When the mode is cancelled or the tool change is executed, the compensation value is changed. When the compensation value is changed in compensation mode, the new compensation is value after the tool change is executed.
8. When the tool compensation programs are execute, errors or alarms occur, but G code remains unchanged, at the moment, pressing reset key directly cancel the tool compensation C state.
9. G33/G34 does not execute the tool nose radius compensation.

## Part Two Programming

Machine the workpiece shown in Fig. 8-4-1, the tool nose radius  $R=1$  and the tool is the first.

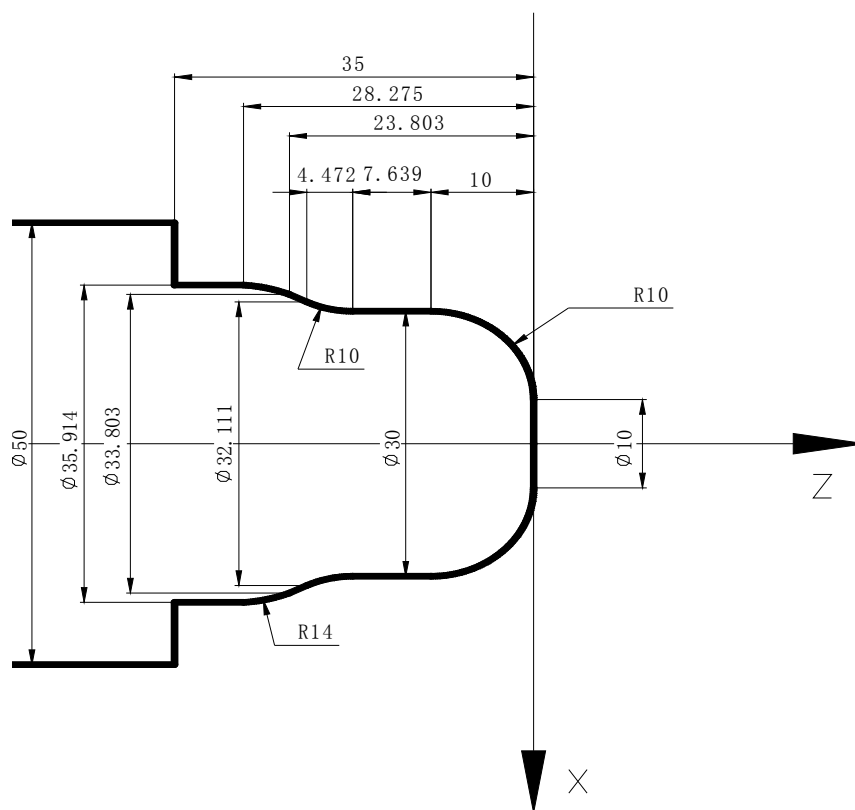


Fig. 8-4-1 tool compensation C example ①

|                        |                                                    |
|------------------------|----------------------------------------------------|
| O0001;                 |                                                    |
| N010 G0 X100.0 Z100.0; | (Positioning to safety position)                   |
| N020 M3 S300;          | (Spindle rotation CCW, speed: 300r/min)            |
| N030 M8;               | (Cooling ON)                                       |
| N040 T0101;            | (Change No. 1 tool, execute its tool compensation) |
| N050 G00 X10.0 Z10.0;  | (rapid positioning, approach workpiece)            |
| N060 G42 G1 Z0 F80;    | (Start executing tool nose radius compensation)    |

```

N070 G3 X30 Z-10 R10;
N080 G1 Z-17.639;
N090 G2 X32.111 Z-22.111 R10;
N100 G1 X33.803 Z-23.803;
N110 G3 X35.914 Z-28.275 R14;
N120 G1 Z-35;
N130 X50;
N140 G40 G0 X80 Z80;      (Cancel tool nose radius compensation)
N150 M09;                  (Cooling OFF)
N160 G00 X100.0 Z100.0 T0200; (Rapid return to safety position, change reference tool,
                                clear tool offset)
N170 M30;                  (End of program)

```

Tool compensation C example ②:

Machine the workpiece shown in Fig. 8-4-2, the tool nose radius  $R=1$  and the tool is the first.

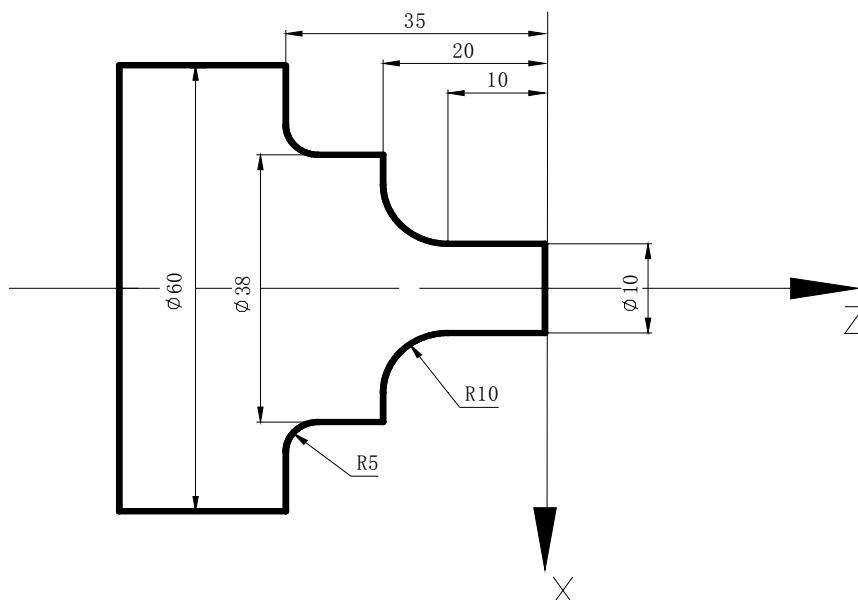


Fig. 8-4-2 tool compensation C example ②

```

O0002
N1 G0 X100 Z100;
N2 M3 S800;
N3 M8;
N4 T0202;
N5 G0 X70 Z10;
N6 G71 U3 R1;
N7 G71 P8 Q14 U0 W0 F120;
N8 G0 X10;
N9 G1 Z-10 F80;
N10 G02 X30 W-10 R10;
N11 G1 X38;

```

```

N12 Z-30;
N13 G02 X48 W-5 R5;
N14 G1 X60;
N15 G0 X100 Z80;
N16 M3 S300;
N17 T0101;
N18 G0 X70 Z10;
N19 G42 G70 P8 Q14;    (G42 and G70 are in the same block to execute the tool nose radius
                        compensation)
N20 G40 G0 X80 Z50;    (Cancel tool nose radius compensation)
N21 G0 X100 Z100 T0200;
N22 M30;

```

Tool compensation C example ③:

Machine the workpiece shown in Fig. 8-4-3, the tool nose radius  $R=1$  and the tool is the first.

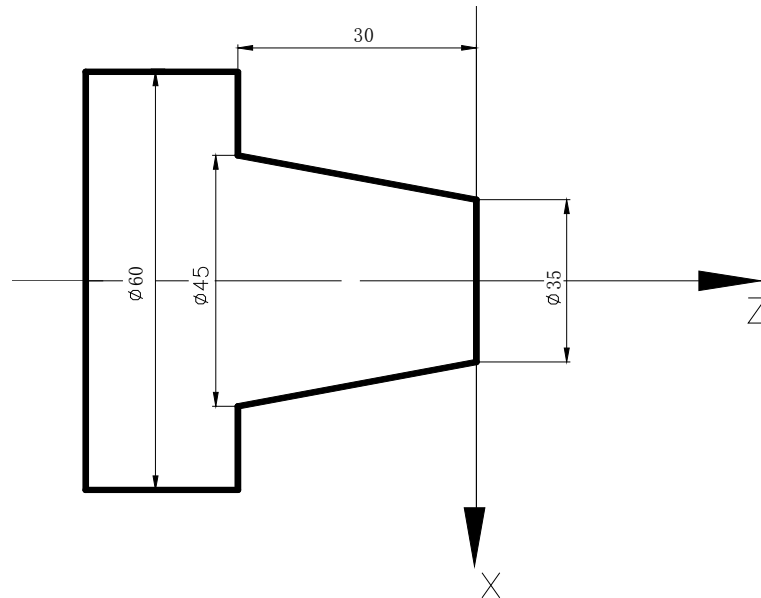


Fig. 8-4-3 tool compensation C example ③

```

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

```

## **PART THREE OPERATION**



## Chapter One Operation Panel

### 1.1 Panel division

GSK980TB3 systems use integrated panel including LCD (liquid crystal display), Edit keypad and machine control display sections as shown in the figure bellow:

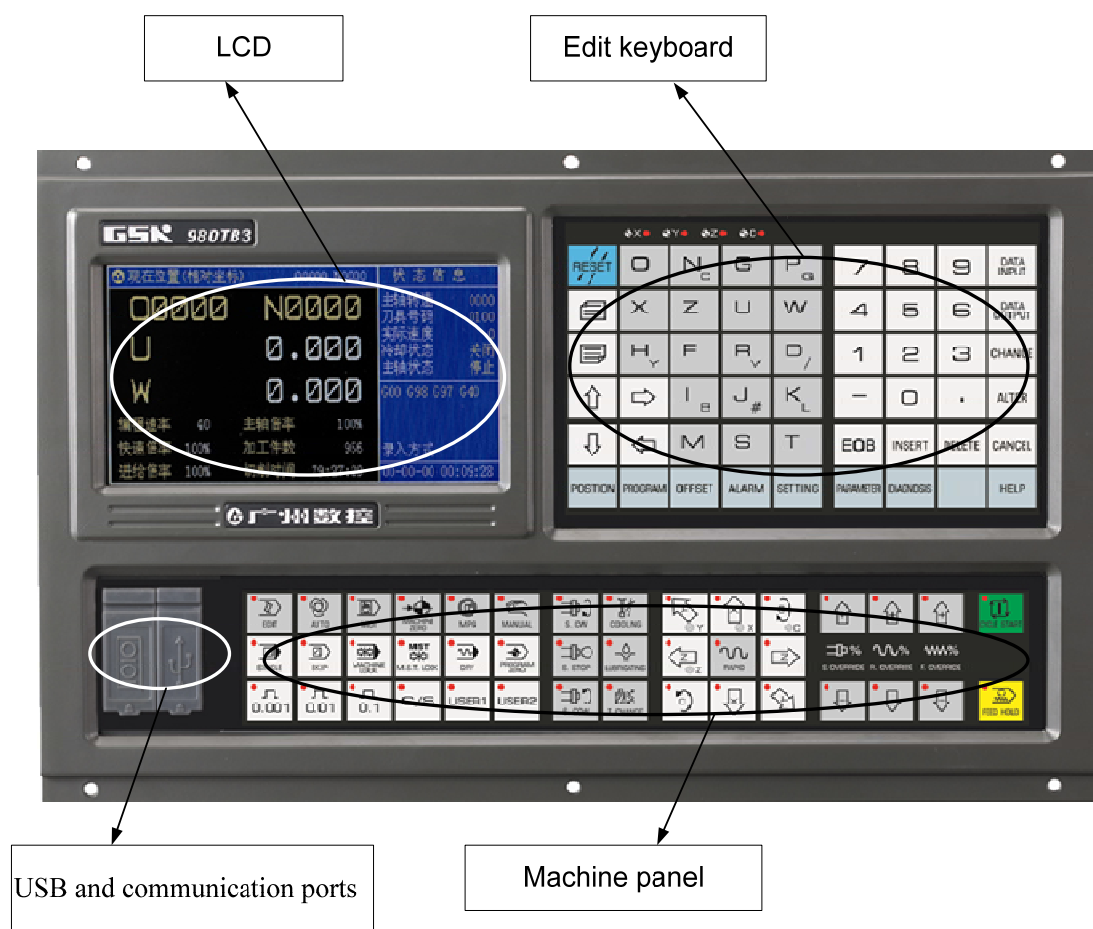


Fig.1-1-1 GSK980TB3

GSK980TB3-V adopts vertical panel structure, which has RS232 and USA communication interface:



Fig.1-1-2 GSK980TB3-V

## 1.2 Instructions for Panel Functions

### 1.2.1 LCD (liquid crystal) display area

The display area of the system adopts color 7-inch liquid crystal display.

### 1.2.2 Edit Keypad Area

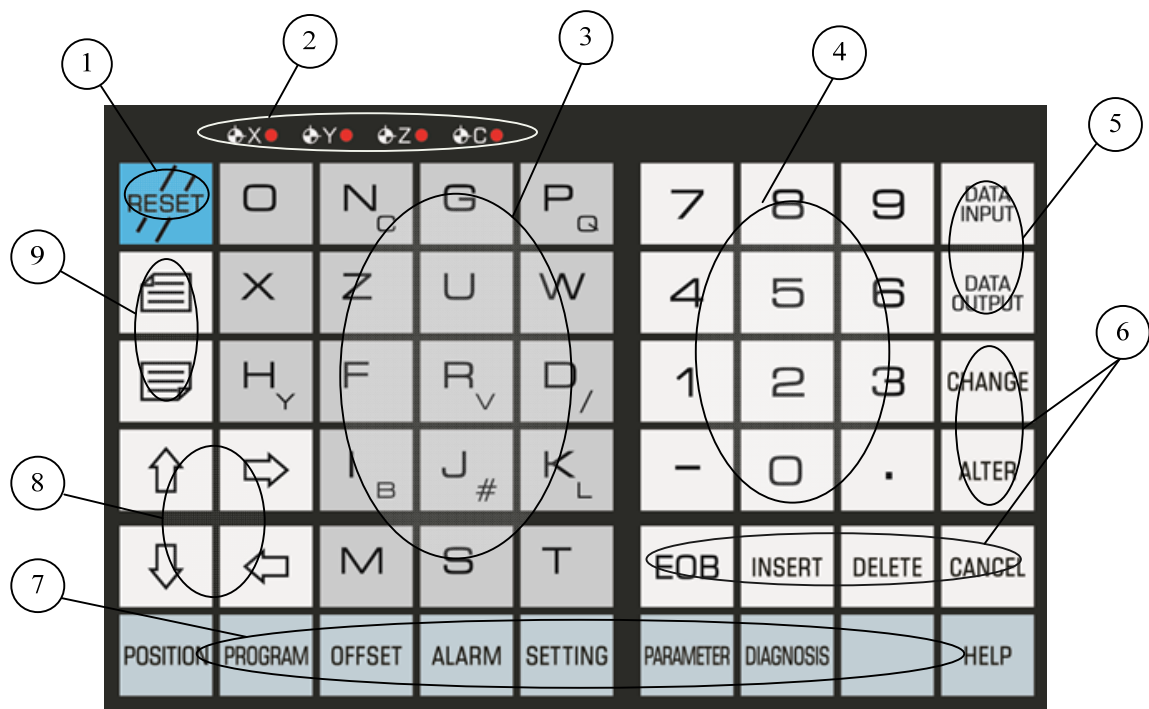


Fig. 1-2-2-1 GSK980TB3

The edit keypad area is further divided into 9 parts. Their detailed usages are as follows:

| No. | Name                 | Function explanation                                                        |
|-----|----------------------|-----------------------------------------------------------------------------|
| 1   | Reset key            | The CNC reset, feed and output stop etc.                                    |
| 2   | Indicator            | Function indicator                                                          |
| 3   | Address key          | Address input                                                               |
| 4   | Numerical key        | Digit input                                                                 |
| 5   | I/O key              | For parameter, offset input. The files input and output by RS232 interface. |
| 6   | Functional key       | Program insertion, alteration and deletion in program edit                  |
| 7   | Screen operation key | Press any key to enter corresponding interface (details are shown below)    |
| 8   | Cursor moving key    | Cursor moving up, down, left or right                                       |
| 9   | Page key             | For page turning in an interface                                            |

1.2.3 Introduction for Screen Operation Keys

There are 8 page display keys on the operation panel of the system 980TB3, see the following figure:

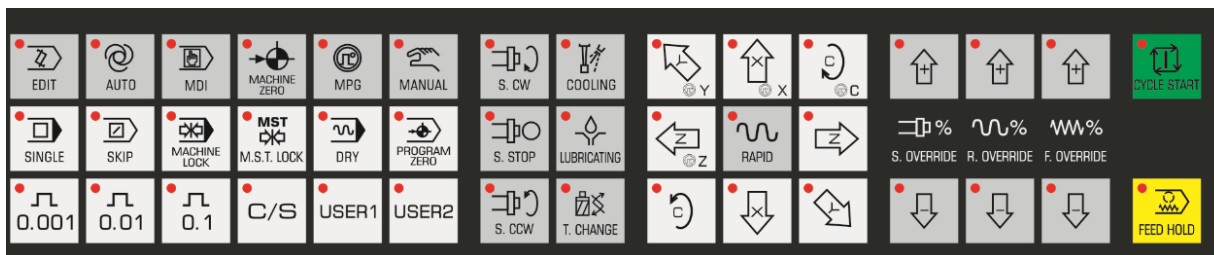


| Name     | Function Explanation            | Remarks                                                                                                                |
|----------|---------------------------------|------------------------------------------------------------------------------------------------------------------------|
| Position | To enter the position interface | The relative coordinate, absolute coordinate, integrated coordinate and the position/program interfaces of the current |

|           |                                  |                                                                                                              |
|-----------|----------------------------------|--------------------------------------------------------------------------------------------------------------|
|           |                                  | point can be switched by pressing the page key.                                                              |
| Program   | To enter the program interface   | The program, MDI and catalogue interfaces can be switched by pressing the page key.                          |
| Offset    | To enter the offset interface    | The contents of number 001~064 and 101~164 can be switched by pressing this key continuously                 |
| Alarm     | To enter the alarm interface     | The interfaces of the alarm message and the external message can be switched by the page key.                |
| Setting   | To enter the setting interface   | The setting and graph interfaces can be switched by pressing this key continuously                           |
| Parameter | To enter the parameter interface | The parameter and screw-pitch compensation interfaces can be switched by pressing this key continuously      |
| Diagnosis | To enter the diagnosis interface | The diagnosis, machine panel and macro variable interfaces can be switched by pressing this key continuously |

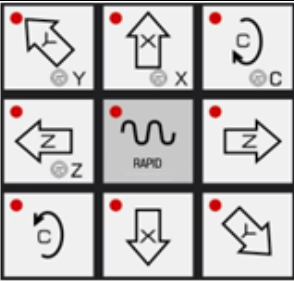
## 1.2.4 Machine Control Area

### 1.2.4.1 Machine Control Area of the 980TB3



| Key                                                                                 | Name              | Function explanations      | Remarks and operation instructions                                                           |
|-------------------------------------------------------------------------------------|-------------------|----------------------------|----------------------------------------------------------------------------------------------|
|  | Edit mode key     | To enter Edit mode         | Switch to Edit mode from Auto mode. CNC decelerates to stop after present block is finished. |
|  | Auto mode key     | To enter Auto mode         | In Auto mode, CNC selects internal memory program                                            |
|  | MDI mode key      | To enter MDI mode          | Switch to MID mode from Auto mode. CNC decelerates to stop after present block is finished   |
|  | Machine zero mode | To enter Machine zero mode | After Auto mode is switched to Machine zero mode, the system immediately decelerates to stop |
|  | MPG               | To enter MPG mode          | After Auto mode is switched to MPG mode, the system immediately decelerates to stop          |
|  | Manual            | Manual mode key            | After Auto mode is switched to, the system immediately decelerates to stop                   |

|                                                                                     |                                         |                                                                                                  |                                                                            |
|-------------------------------------------------------------------------------------|-----------------------------------------|--------------------------------------------------------------------------------------------------|----------------------------------------------------------------------------|
|    | Single block                            | For switching of block/ blocks.<br>Single block indicator lights up if Single mode is active     | This mode is active in Auto mode and MDI mode                              |
|    | Block Skip key                          | For skipping of block headed with “/” sign. If the indicator lights up, the block is skipped     | This mode is active in Auto mode and MDI mode                              |
|    | Dry run                                 | If dry run is active, the indicator lights up                                                    | This mode is active in Auto mode and MDI mode                              |
|    | M.S.T Lock key                          | If the M.S.T. lock is set for ON, the indicator lights up. and M,S,T function output is inactive | This mode is active in Auto mode and MDI mode                              |
|    | Machine lock key                        | If the Machine lock is set for ON, the indicator lights up. and axis output is inactive          | This mode is active in Auto mode, MDI, Machine zero, MPG, Manual mode      |
|   | Spindle control keys                    | Spindle CW<br>Spindle stop<br>Spindle CCW                                                        | In Manual mode, they are active when the spindle is in analog control mode |
|  | Spindle override key                    | For spindle speed adjustment (It is active in spindle analog control)                            | Any mode                                                                   |
|  | Rapid override                          | Rapid override key                                                                               | This mode is active in Auto mode, MDI, Machine zero, MPG, Manual mode      |
|  | Feedrate override key                   | For adjustment of the federate                                                                   | This mode is active in Auto mode, MDI, Machine zero, MPG, Manual mode      |
|  | Manual step, MPG override selection key | For selection of manual step, MPG override.                                                      | This mode is active in MPG and Manual mode                                 |
|  | MPG axis selection key                  | Corresponding axis selection for manual pulse generator (MPG)                                    | It is active in MPG mode                                                   |
|  | Lubricating on/off key                  | For lubricating on/off                                                                           | This mode is active in Auto mode, MDI, MPG, Manual mode                    |
|  | Cooling on/off key                      | For cooling on/off                                                                               | This mode is active in Auto mode, MDI, MPG, Manual mode                    |
|  | Work light on/off key                   | For work light on/off                                                                            | This mode is active in MPG, Manual mode                                    |

|                                                                                   |                                              |                                                                                           |                                                                              |
|-----------------------------------------------------------------------------------|----------------------------------------------|-------------------------------------------------------------------------------------------|------------------------------------------------------------------------------|
|  | <p>Manual feed key</p> <p>Rapid traverse</p> | <p>For positive/negative moving of X, Y, Z axes in manual mode. Rapid traverse on/off</p> | <p>This mode is active in machine zero, manual mode, MPG</p> <p>Any mode</p> |
|  | <p>Feed hold key</p>                         | <p>Auto operation of the system is stopped by pressing this key</p>                       | <p>This mode is active in Auto mode, MDI mode</p>                            |
|  | <p>Cycle start key</p>                       | <p>The system operates automatically by pressing this key</p>                             | <p>This mode is active in Auto mode, MDI mode</p>                            |

## Chapter Two System Power ON/OFF and Safety Operation

### 2.1 System Power-on

Before this GSK980MDa is powered on, confirm that:

1. The machine is in a normal state.
2. The power voltage conforms to the requirement of the machine.
3. The connection is correct and secure.

The current position (Absolute POS) is displayed after the self-check and initialization are finished.

| POSITION (RELATIVE) |                   | Message         | 09:03 |
|---------------------|-------------------|-----------------|-------|
| <b>O0002</b>        | <b>N0000</b>      | Axis Rev        | 0000  |
| <b>U</b>            | <b>0.000</b>      | Tool No         | 0100  |
| <b>W</b>            | <b>0.000</b>      | Fact Rate       | 0     |
|                     |                   | Cool Mes        | OFF   |
|                     |                   | Axis Mes        | Stop  |
|                     |                   | G00 G98 G97 G40 |       |
| FEEDRATE 40         | SPDLRIDE 100%     | MDI             |       |
| RAPDRIDE 100%       | WORK NO. 0        |                 |       |
| OVERRIDE 100%       | RUN TIME 00:00:00 |                 |       |

Fig.2-1-1

### 2.2 System Power-off

Before power is off, confirm that:

- 1 X,Z axes of the CNC are stopped;
- 2 Miscellaneous functions (spindle, pump etc.) are off;
- 3 Cut off CNC power prior to machine power cutting off.

Before power is off, check that:

- 1 LED indicating cycle start of the panel is in a halted state;
- 2 All movable parts on CNC machine are in a halted state;
- 3 CNC power is cut off by pressing POWER OFF button.

#### Emergency Power-off

Under the emergencies during the machine operation, the machine power should be cut off immediately to avoid the incidents. However, it should be noted that there may be an error between the system displayed coordinate and the actual position. Therefore, the machine zero and tool setting operations should be performed again.

**Note:** Please see the machine builder's manual for the machine power-off operation.

## 2.3 Safety Operation

### 2.3.1 Reset



Press  key to the reset CNC if there are abnormal output and axis action about it:

1. All axes movement stops
2. Auto operation ends, modal function and state are held on.
3. By modifying parameter **N0:02#6**, it is able to set offset vector is unchanged/ deleted during reset.
4. By modifying parameter **N0:12#0**, it is able to set whether M03, M04, lubricating and cooling is off or not after reset.
5. By modifying parameter **N0:13#1**, it is able to set whether the macro common variable is deleted/ unchanged during reset.
6. By modifying parameter **N0:13#3**, it is able to set whether the expansion output IO port is off or not during reset.

### 2.3.2 Emergency Stop

By pressing the emergency stop button in machine operation, the movement of the machine such as spindle rotation, cooling, etc. are cut off immediately. Release the emergency stop button to leave the emergency stop state.

**Note 1: Make sure that the troubles are removed before the emergency button is eliminated.**

**Note 2: After the button is released, reference return operation should be done to ensure the correctness of the coordinate position.**

**Note 3: Electric shock can be reduced by pressing down the emergency stop button before the system power ON/OFF. Emergency stop signal is a normally-closed contact signal. When the contact is disconnected, the system enters emergency stop state, and the machine stops.**

### 2.3.3 Feed Hold



 key can be pressed to make the running pause during the machine operation. It calls for special notice that the feed hold state can be entered only when the next block is not thread cutting or cycle command operation after the current command is performed in these operations.

### 2.3.4 Power-off

Under the emergencies during the machine operation, the machine power should be cut off immediately to avoid the accidents. However, it should be noted that there may be an error between the system displayed coordinate and the actual position. Therefore, the tool setting operation should be performed again.

## 2.4 Cycle Start and Feed Hold

Cycle start and feed hold keys on panel are used for program start and pause operation in Auto and MDI mode.

The additional panel can connect to the cycle start and feed hold keys.

By modifying parameter **NO:14#0**, **NO:14#1**, it is able to set whether the external start and pause are shielded.

## 2.5 Overtravel Protection

Overtravel protection should be employed to prevent the damage to the machine due to overtravel of the feed axis.

### 2.5.1 Hardware Overtravel Protection

The stroke limit switches are fixed at the positive and negative maximum stroke position of X and Z axes respectively. If the overtravel occurs, the running axis decelerates to eventually stop when it contacts with the stroke switch, and the emergency alarm is issued.

#### Detailed explanations

Once the workbench presses the limit switch during the operation, the system decelerates immediately to stop all operation of axes. The control signals of spindle, lubricating and cooling are off and overtravel alarm occurs.

#### Steps for eliminating the hardware overtravel alarm

1. Press and hold the overtravel release button, now the system alarm is eliminated.
2. Move the carriage at the opposite side (move out negatively for positive overtravel, vice versa) to leave the limit switch.
3. Release the overtravel limit button.

### 2.5.2 Software Overtravel Protection

Software stroke range is set by parameter P46~ P49 on the data parameter (appendix 1). Set coordinate value of the machine as reference value. If the machine position (machine coordinate) exceeds software stroke range, overtravel alarm will occur. When the overtravel alarm occurs, move the axis reversely in the manual mode and then press the reset key to eliminate the alarm.

## Chapter Three Interface Display and Data Modification and Setting

### 3.1 Position Display

#### 3.1.1 Four Ways for Interface Display

Press  key to enter Position Interface. There are four modes in this interface such as **【REL.】**, **【ABS.】**, **【COM.】**, **【POSC.】**. They can be viewed by pressing  or  key. Detailed information of each interface is as follows:

1) Relative POS.: It displays the current tool position in relative coordinate system (see Fig. 3-1-1-1)

|                                                                                                                   |                   |                 |       |
|-------------------------------------------------------------------------------------------------------------------|-------------------|-----------------|-------|
|  POSITION (RELATIVE) 00001 N0000 |                   | Message         | 09:03 |
| <b>O0001 N0000</b>                                                                                                |                   | Axis Rev        | 0000  |
| <b>U 0.000</b>                                                                                                    |                   | Tool No         | 0100  |
| <b>W 0.000</b>                                                                                                    |                   | Fact Rate       | 0     |
|                                                                                                                   |                   | Cool Mes        | OFF   |
|                                                                                                                   |                   | Axis Mes        | Stop  |
|                                                                                                                   |                   | G00 G98 G97 G40 |       |
| FEEDRATE 40                                                                                                       | SPDLRIDE 100%     | MDI             |       |
| RAPDRIDE 100%                                                                                                     | WORK NO. 0        |                 |       |
| OVERRIDE 100%                                                                                                     | RUN TIME 00:00:00 |                 |       |

Fig. 3-1-1-1

Steps for clearing the relative coordinate system: Coordinate system U axis is blinking by pressing



key, and then press  key to cancel the coordinate system. The operation of the W axis is the same as that of the U axis.

Note: The relative POS. will be cleared automatically at power off.

2) Absolute POS.: It displays the current tool position in absolute coordinate system (see Fig. 3-1-1-2)

|                                                                                                                     |                   |                 |       |
|---------------------------------------------------------------------------------------------------------------------|-------------------|-----------------|-------|
|  POSITION (ABSOLUTE) 00001 N0000 |                   | Message         | 09:03 |
| <b>O0001 N0000</b>                                                                                                  |                   | Axis Rev        | 0000  |
| <b>X 0.000</b>                                                                                                      |                   | Tool No         | 0100  |
| <b>Z 0.000</b>                                                                                                      |                   | Fact Rate       | 0     |
|                                                                                                                     |                   | Cool Mes        | OFF   |
|                                                                                                                     |                   | Axis Mes        | Stop  |
|                                                                                                                     |                   | G00 G98 G97 G40 |       |
| FEEDRATE 40                                                                                                         | SPDLRIDE 100%     | MDI             |       |
| RAPDRIDE 100%                                                                                                       | WORK NO. 0        |                 |       |
| OVERRIDE 100%                                                                                                       | RUN TIME 00:00:00 |                 |       |

Fig. 3-1-1-2

3) Integrated interface: The following coordinate position values are displayed:

- (A) Position on relative coordinate system
- (B) Position on absolute coordinate system
- (C) Position on machine coordinate system
- (D) Distance-to-go (Display in Auto and MDI mode)

Display page is as follows (Fig.3-1-1-3):

|                                                                                            |               |                 |       |
|--------------------------------------------------------------------------------------------|---------------|-----------------|-------|
|  POSITION | 00001 N0000   | Message         | 09:03 |
| (RELATIVE)                                                                                 | (ABSOLUTE)    | Axis Rev        | 0000  |
| U 2.563                                                                                    | X 301.059     | Tool No         | 0100  |
| W 3.587                                                                                    | Z 0.684       | Fact Rate       | 0     |
|                                                                                            |               | Cool Mes        | OFF   |
|                                                                                            |               | Axis Mes        | Stop  |
| (MACHINE)                                                                                  | (DISTANCE GO) | G00 G98 G97 G40 |       |
| X 400.256                                                                                  | X 0.000       |                 |       |
| Z -100.236                                                                                 | Z 0.000       |                 |       |
|                                                                                            |               | MDI             |       |

Fig. 3-1-1-3

4) Program monitoring mode: In this interface, relative coordinate and absolute coordinate of the current position and the current block can be displayed at the same time.

|                                                                                              |             |                 |       |
|----------------------------------------------------------------------------------------------|-------------|-----------------|-------|
|  POSITION | 00001 N0000 | Message         | 09:04 |
| (RELATIVE)                                                                                   | (ABSOLUTE)  | Axis Rev        | 0000  |
| U 2.563                                                                                      | X 301.059   | Tool No         | 0100  |
| W 3.587                                                                                      | Z 0.684     | Fact Rate       | 0     |
|                                                                                              |             | Cool Mes        | OFF   |
|                                                                                              |             | Axis Mes        | Stop  |
| 00001;                                                                                       |             | G00 G98 G97 G40 |       |
| G50 X250. Z450. ;                                                                            |             |                 |       |
| T0101;                                                                                       |             |                 |       |
| G00 X100. Z200. ;                                                                            |             |                 |       |
| G90 U-10. W-50. R-1.5 F500. ;                                                                |             |                 |       |
|                                                                                              |             | MDI             |       |

Fig. 3-1-1-4

### 3.1.2 Display of Machining Time, Parts Number, Programming Speed, Override and Actual Speed

In absolute and relative interfaces of the POS., programming rate, rapid override, federate override, spindle override, parts number, cutting time can be displayed. Spindle speed, tool number, cooling state, spindle state, G function code, operation mode, the current date and time etc. can be displayed in state message column. See the Fig. 3-1-2-1.

|                     |                   |                 |       |
|---------------------|-------------------|-----------------|-------|
| POSITION (RELATIVE) |                   | Message         | 09:03 |
| O0002               | N0000             | Axis Rev        | 0000  |
| U                   | 0.000             | Tool No         | 0100  |
| W                   | 0.000             | Fact Rate       | 0     |
|                     |                   | Cool Mes        | OFF   |
|                     |                   | Axis Mes        | Stop  |
|                     |                   | G00 G98 G97 G40 |       |
| FEEDRATE 40         | SPDLRIDE 100%     | MDI             |       |
| RAPDRIDE 100%       | WORK NO. 0        |                 |       |
| OVERRIDE 100%       | RUN TIME 00:00:00 |                 |       |

Fig. 3-1-2-1

Detailed significances are as follows:

Programming rate: the rate specified by F code.

Rapid override: current rapid override of the system.

Feedrate override: current federate override of the system.

Spindle override: Spindle speed override for adjusting spindle speed.

Parts number: add 1 when M30 (or M99 in the main program) is executed.

Cutting time: It begins timing when auto operation is started. Unit: hour, minute, second.

Spindle speed: actual speed from the spindle encoder.

Tool number: tool number and tool offset number of the current tool.

Actual speed: actual processing speed after the magnification is multiplied.

Cooling state: cooling on/off state.

Spindle state: spindle CCW/STOP/CW.

G function code: modal G code of the current system.

### 3.1.3 Clearing Methods of Parts Number and Cutting Time

- 1) Switch to the position interface.

- 2) By pressing  key, parts number and cutting time on the interface are blinking. Then press  key, parts number and cutting time are cancelled. If user presses other keys while they are blinking, the value is not change.

**Note 1:** The encoder shall be installed on the spindle for displaying the actual spindle speed.

**Note 2:** Actual rate = F value of the programming rate × magnification. In G00, speed of each axis is set by data parameter P21~P22. It can be adjusted by the rapid override. The upper cutting speed is set by data parameter P27. If it exceeds the upper value, the system will alarm.

**Note 3:** By setting the parameter NO:3#6, it is able to set whether the power-off memory is active for the parts number and the cutting time.

3.2 Program Display

Press  key to enter program display interface. There are three modes in this interface such as **【PROG.】** , **【MDI input】** , **【PRG content】** . They can be viewed by pressing  or  key. Detailed information of each interface is as follows:

- 1) Program display: the current program block in memory unit is displayed in this interface (see Fig. 3-2-1)

|                                                                                                                                                                                               |             |                                                                               |       |
|-----------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|-------------|-------------------------------------------------------------------------------|-------|
|  PROGRAM                                                                                                     | 00001 N0000 | Message                                                                       | 09:04 |
| 00001;<br>G50 X250. Z450. ;<br>T0101;<br>G00 X100. Z200. ;<br>G90 U-10. W-50. R-1.5 F500. ;<br>G90 U-50. W-10. R-3. F350. ;<br>G00 X120. ;<br>T0202;<br>M03 S01;<br>G92 X80. W-65. R5.1 F0.5; |             | Axis Rev 0000<br>Tool No 0100<br>Fact Rate 0<br>Cool Mes OFF<br>Axis Mes Stop |       |
| ADRS Row 2                                                                                                                                                                                    |             | G00 G98 G97 G40                                                               |       |
|                                                                                                                                                                                               |             | EDIT                                                                          |       |

Fig. 3-2-1

The detailed program edit, please see the chapter 10 operation of editing on operation manual. Only in program display mode of the edit mode, all contents of the current program can be viewed and

edited by pressing  or  key.

- 2) MDI input display: multi-block can be edited and executed in the edit mode. Its program format is the same as that of the edit mode.

MDI operation applies to simple testing program (see fig. 3-2-2).

|                                                                                                           |             |                                                                               |       |
|-----------------------------------------------------------------------------------------------------------|-------------|-------------------------------------------------------------------------------|-------|
|  PROGRAM               | 00001 N0000 | Message                                                                       | 10:04 |
| X<br>Z G94 F 1000.000<br>U G97 M<br>W G98 S 0<br>R T 0100<br>F<br>M G21<br>S G40<br>T<br>P<br>Q SACT 0000 |             | Axis Rev 0000<br>Tool No 0100<br>Fact Rate 0<br>Cool Mes OFF<br>Axis Mes Stop |       |
| ADRS                                                                                                      |             | G00 G98 G97 G40                                                               |       |
|                                                                                                           |             | MDI                                                                           |       |

Fig. 3-2-2

- 3) The program (catalogue) displays catalogue display interface. See fig. 3-2-3 for its contents.
- (a) System version number: current version of the system used.
  - (b) Number of the stored programs: number of the stored programs (including subprogram)

Available space: the space can be used.

- (c) Used storage capacity: the space occupied by the stored programs (displayed by character number).

Available space: the space can be used.

- (d) Program catalogue: the numbers of the stored programs are displayed in order. The programs are displayed in the order of the size of the program name.

|                                                                                           |             |                 |       |
|-------------------------------------------------------------------------------------------|-------------|-----------------|-------|
|  PROGRAM | 00001 N0000 | Message         | 10:04 |
| SYSTEM EDIT:GSK-980TB3(08) V3.74.083                                                      |             | Axis Rev        | 0000  |
| PRG. NO. USED: 13 FREE:687                                                                |             | Tool No         | 0100  |
| MEMORY USED: 704 FREE: 23840                                                              |             | Fact Rate       | 0     |
| PRG. LIBRARY:                                                                             |             | Cool Mes        | OFF   |
| 00000 00001 00002 00003 00004 00005                                                       |             | Axis Mes        | Stop  |
| 00006 00007 00008 00009 00010 00011                                                       |             | G00 G98 G97 G40 |       |
| 00045                                                                                     |             |                 |       |
| ADRS                                                                                      |             | MDI             |       |

Fig. 3-2-3

### 3.3 Display, Modification and Setting of the Tool Offset

#### 3.3.1 Tool Offset Display

Press  key to enter offset Interface. There are numbers 001~064 and 101~164 two parts in this

interface. They can be viewed by pressing  key continuously. Detailed information is as follows:

- 1) Offset interface The offset interface is as follows (see fig. 3-3-1-1):

|                                                                                                 |             |       |         |   |                 |       |
|-------------------------------------------------------------------------------------------------|-------------|-------|---------|---|-----------------|-------|
|  OFFSET (mm) | 00001 N0000 |       |         |   | Message         | 10:09 |
| No.                                                                                             | X           | Z     | R       | T | Axis Rev        | 0000  |
| 000                                                                                             | 0.000       | 0.000 | 0.000   | 0 | Tool No         | 0100  |
| 001                                                                                             | 0.000       | 0.000 | 0.000   | 0 | Fact Rate       | 0     |
| 002                                                                                             | 0.000       | 0.000 | 0.000   | 0 | Cool Mes        | OFF   |
| 003                                                                                             | 0.000       | 0.000 | 0.000   | 0 | Axis Mes        | Stop  |
| 004                                                                                             | 0.000       | 0.000 | 0.000   | 0 | G00 G98 G97 G40 |       |
| 005                                                                                             | 0.000       | 0.000 | 0.000   | 0 |                 |       |
| 006                                                                                             | 0.000       | 0.000 | 0.000   | 0 |                 |       |
| 007                                                                                             | 0.000       | 0.000 | 0.000   | 0 | MDI             |       |
| ABSO X                                                                                          | 100.000     | Z     | 100.000 |   |                 |       |
| RELA U                                                                                          | -169.380    | W     | 122.337 |   |                 |       |
| ADRS                                                                                            |             |       |         |   |                 |       |

Fig. 3-3-1-1

There are numbers 001~064 and 101~164 two parts in tool offset interface. Number 001~064 are used to modify tool offset value, while number 101~164 are used to set tool offset value (see Tool Setting for detailed functions). Number 000 and 100 are used to move work piece coordinate in parallel direction. X and Z are tool offset value from the tool post center to the tool nose along the direction of

X, Z axis respectively. R is the radius compensation value of the imaginary tool nose. T is the number of the imaginary tool nose (See C Tool Offset Function Instructions for detailed functions).

### 3.3.2 Tool Offset Modification and Setting

In the offset interface, methods of tool offset setting are as follows:

- 1) Press  to enter the offset interface.
- 2) Move the cursor to the position where the offset number will be input.

Method 1: Press the page key to enter the page where the offset value to be modified locates. By pressing the direction keys, the cursor is moved to the position where the offset to be modified locates.

Method 2: Press  key to input the number to be altered. Then press  key, the cursor moves to the input number directly. Finally press  key to return to the state of the data input.

- 3) Press  or  key to select the tool offset axis, an input the offset value. After pressing  key, the system will input the offset value automatically, and it will be displayed on the LCD screen.

**Note 1:** The offset value can not be set in auto operation.

**Note 2:** When you adjust the tool offset value in the range 001~064, use U, W to input the adjusting value.

When tool setting is performed in the range 101~164, input actual measurement value of the X, Z

respectively. If the add and subtraction of the current offset value is required, press  key for

the X axis, and press  key for Y axis, then, input the value to be added or subtracted, finally

press  key. For example, the offset value of Z axis is 3, then input W5 to get Z axis offset value 3+5=8.

### 3.4 Alarm Display

Press  key to enter alarm Interface. There are two modes in this interface such as 【alarm signal】,

【external information】. They can be viewed by pressing  or  key. Detailed information of each interface is as follows:

1. For the alarm interface, see Fig. 3-4-1:

|                                                                                                                                |           |       |
|--------------------------------------------------------------------------------------------------------------------------------|-----------|-------|
|  ALARM 00001 N0000<br><br>No alarm infomation | Message   | 10:09 |
|                                                                                                                                | Axis Rev  | 0000  |
|                                                                                                                                | Tool No   | 0100  |
|                                                                                                                                | Fact Rate | 0     |
|                                                                                                                                | Cool Mes  | OFF   |
|                                                                                                                                | Axis Mes  | Stop  |
| G00 G98 G97 G40                                                                                                                |           |       |
| MDI                                                                                                                            |           |       |

Fig. 3-4-1

The system will enter the alarm interface automatically when troubles occur. The current P/S alarm number and detailed alarm contents are displayed in the alarm interface, see appendix 2 for the contents.

2. For the external information interface, see Fig. 3-4-2:

|                                                                                                        |           |       |
|--------------------------------------------------------------------------------------------------------|-----------|-------|
|  EXTERIOR 00001 N0000 | Message   | 10:09 |
|                                                                                                        | Axis Rev  | 0000  |
|                                                                                                        | Tool No   | 0100  |
|                                                                                                        | Fact Rate | 0     |
|                                                                                                        | Cool Mes  | OFF   |
|                                                                                                        | Axis Mes  | Stop  |
| G00 G98 G97 G40                                                                                        |           |       |
| MDI                                                                                                    |           |       |

Fig. 3-4-2

### 3.5 Setting Display

Press  key to enter the setting interface that includes **【Setting】** , **【Graph】** two types of display modes. They can be viewed and modified by pressing  key repeatedly.

#### 3.5.1 Instruction for **【Setting】** Interface Operation

**【Setting】** interface includes two interfaces **【Setting】** and **【User Password】** , which can be viewed by

pressing  or  key.

1. See the following figure for the setting interface (see 3-5-1-1):

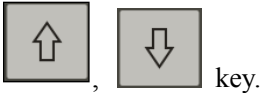
|                                                                                           |             |                 |       |
|-------------------------------------------------------------------------------------------|-------------|-----------------|-------|
|  SETTING | 00001 N0000 | Message         | 10:13 |
| <u>_</u> PARM SWT: *OFF ON                                                                |             | Axis Rev        | 0000  |
| PROG KEY: OFF *ON                                                                         |             | Tool No         | 0100  |
| Datetime set: 00-00-00 00:00:00                                                           |             | Fact Rate       | 0     |
|                                                                                           |             | Cool Mes        | OFF   |
|                                                                                           |             | Axis Mes        | Stop  |
|                                                                                           |             | G00 G98 G97 G40 |       |
|                                                                                           |             | MDI             |       |

Fig. 3-5-1-1

Operating instructions:

On the above figure, the parameter switch and the program switch ON/OFF and time can be set. The detailed operation methods and steps are as follows:

- (a) Move the cursor to the position of parameter switch, program switch or time setting by pressing



- (b) Press  key to set parameter switch or program switch to ON. At this time, parameter modification, setting and program editing can be performed. The system displays number P/S100 alarm when the parameter switch is set to ON. Meanwhile, input parameter and then return to the

setting interface. Press  key to set parameter switch or program switch to OFF, and the alarm is eliminated automatically after the parameter switch is off.

- (c) In <MDI mode>, move the cursor to time setting, and key in data, time by sequence. Then press

 key to confirm it, if input data is wrong, input it again or press  key to cancel it word by word from the back.

2. See the following figure for the user password interface (see fig3-5-1-2)

|                                                                                                            |                                                                               |
|------------------------------------------------------------------------------------------------------------|-------------------------------------------------------------------------------|
|  USERPASSWORD 00001 N0000 | Message 10:13                                                                 |
| — Input Pswd: — (Lev: System)                                                                              | Axis Rev 0000<br>Tool No 0100<br>Fact Rate 0<br>Cool Mes OFF<br>Axis Mes Stop |
|                                                                                                            | G00 G98 G97 G40                                                               |
|                                                                                                            | MDI                                                                           |

Fig. 3-5-1-2

Operating instructions:

- (a) Enter this interface in MDI mode, press  key to select the password level. There are 4 levels, which are user, system, manufacturer and LMT.

- (b) After selecting the level, input level password and press  key to do confirmation. If the operation is right, it enters to password modification interface and the password can be modified. The password consists of 1~6 numbers. If the current level is User or LMT, the system prompts: password is correct. you can modify data parameter and screw-pitch compensation data. Otherwise, it prompts: the input password is wrong. When the current level is manufacturer or system, the system prompts: password is correct. You can modify data parameter and screw-pitch compensation data. Otherwise, it prompts: the input password is wrong.

**Note: The password is not always numbers, it includes “-” and “.”.**

- (c) After modification, input the data again if wrong password is input or original input password is inactive.

### 3.5.2 Operating Instructions for 【Graph】 Interface

【Graph】 interface includes two interfaces 【Graph】 and 【Drafting】 , which can be viewed by pressing



or



key.

1. See the following figure for the 【Graph】 interface (see 3-5-2-1):

|                                                                                                     |             |                 |       |
|-----------------------------------------------------------------------------------------------------|-------------|-----------------|-------|
|  Graphic parameter | 00001 N0000 | Message         | 10:04 |
| Coordinat = 1 (XZ:0 ZX:1)                                                                           |             | Axis Rev        | 0000  |
| Scalerate = 2.000                                                                                   |             | Tool No         | 0100  |
| Center X = 0.000 (X Absolute)                                                                       |             | Fact Rate       | 0     |
| Center Z = 0.000 (Z Absolute)                                                                       |             | Cool Mes        | OFF   |
| X Maximum = 100.000                                                                                 |             | Axis Mes        | Stop  |
| Z Maximum = 100.000                                                                                 |             | G00 G98 G97 G40 |       |
| X Minimum = -100.000                                                                                |             |                 |       |
| Z Minimum = -100.000                                                                                |             |                 |       |
| No. 001                                                                                             |             | MDI             |       |

Fig. 3-5-2-1

Operation instructions:

1) . Graph (parameter) interface, see Fig. 3-5-2-1.

A. Significances of Graph (parameter)

Coordinate selection: There are 2 options for setting drafting plane, see the figure:

Set 0 for XZ coordinate, and set 1 for ZX coordinate, see figure 3-5-2-2.

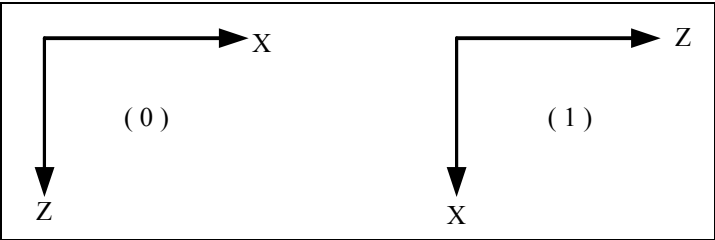


Fig. 3-5-2-2

Scaling ratio: To set drafting scale.

Graph center: To set the workpiece coordinate value to the LCD center in workpiece coordinate system.

Maximum and minimum value: After maximum and minimum values for the displayed axis are set, CNC system scales automatically and center value of the graph is set automatically.

X max.: X axial max. value in graph display (unit: mm)

X min.: X axial min. value in graph display (unit: mm)

Z max.: Z axial max. value in graph display (unit: mm)

Z min.: Z axial min. value in graph display (unit: mm)

B. Setting methods for graph (parameter)

- a. In <MDI mode>, move the cursor to the parameter to be set.
- b. Key in corresponding value according to the actual requirements.

c. Press  key to confirm it.

2) . See the following figure for the **【Drafting】** interface (see 3-5-2-3):

|                                                                                                                                                                                       |             |         |                 |       |
|---------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|-------------|---------|-----------------|-------|
|  PLOT <div>  </div> | 00001 N0000 |         | Message         | 10:04 |
|                                                                                                                                                                                       | X           | 120.000 | Axis Rev        | 0000  |
|                                                                                                                                                                                       | Z           | 205.000 | Tool No         | 0300  |
|                                                                                                                                                                                       |             |         | Fact Rate       | 0     |
|                                                                                                                                                                                       |             |         | Cool Mes        | OFF   |
|                                                                                                                                                                                       |             |         | Axis Mes        | Stop  |
|                                                                                                                                                                                       |             |         | G00 G98 G97 G40 |       |
| *S:Start T:Stop                                                                                                                                                                       |             |         | MDI             |       |

Fig. 3-5-2-3

In drafting interface, processing path of the program can be monitored.

- A. Press  key to enter the state of drafting and mark ‘\*’ moves to **S: drafting start**. In the operation of Auto/MDI/Manual mode, absolute coordinate value is changed and the corresponding motion path is drafted.
- B. Press  key to enter the state of drafting stop and mark ‘\*’ moves to **T: drafting stop**.
- C. Press  key to cancel the drafted graph.
- D. Press up, down, left, right keys to move the whole graph to the appropriate position.

## 3.6 Display, Modification and Setting of the Parameter

### 3.6.1 Parameter Display

Press  key to enter parameter Interface. There are two kinds of display modes, which are **【Parameter】** , **【Screw-pitch compensation parameter】** . They can be viewed by pressing  key continuously.

1) . See the figure for parameter interface (see 3-6-1-1):

|                                                                                                         |          |     |          |                 |       |
|---------------------------------------------------------------------------------------------------------|----------|-----|----------|-----------------|-------|
|  PARAMETER 00001 N0000 |          |     |          | Message         | 10:04 |
| No.                                                                                                     | DATA     | No. | DATA     | Axis Rev        | 0000  |
| 001                                                                                                     | 00000000 | 009 | 00001100 | Tool No         | 0100  |
| 002                                                                                                     | 01000001 | 010 | 00000000 | Fact Rate       | 0     |
| 003                                                                                                     | 01000100 | 011 | 00000001 | Cool Mes        | OFF   |
| 004                                                                                                     | 00100000 | 012 | 00000000 | Axis Mes        | Stop  |
| 005                                                                                                     | 01000000 | 013 | 00000000 | G00 G98 G97 G40 |       |
| 006                                                                                                     | 00000000 | 014 | 10001111 |                 |       |
| 007                                                                                                     | 00000000 | 015 | 00000000 |                 |       |
| 008                                                                                                     | 00000000 | 016 | 00000000 |                 |       |
| LAN SCW *** MDSP WHLA RAD *** ***                                                                       |          |     |          |                 |       |
| Bit0: Unused                                                                                            |          |     |          |                 |       |
| No. 001                                                                                                 |          |     |          | MDI             |       |

Fig. 3-6-1-1

Detailed explanations for the parameter where the cursor locates are displayed in the lower part of the screen. See appendix 1 Parameter Explanation for definitions of each parameter.

2) . See the following figure for the screw pitch compensation interface (See 3-6-1-2):

|                                                                                                                        |        |        |                 |       |
|------------------------------------------------------------------------------------------------------------------------|--------|--------|-----------------|-------|
|  WORMCOMPENSATE (mm/1000) 00001 N0000 |        |        | Message         | 10:04 |
| No.                                                                                                                    | X (um) | Z (um) | Axis Rev        | 0000  |
| 000                                                                                                                    | 0      | 0      | Tool No         | 0100  |
| 001                                                                                                                    | 0      | 0      | Fact Rate       | 0     |
| 002                                                                                                                    | 0      | 0      | Cool Mes        | OFF   |
| 003                                                                                                                    | 0      | 0      | Axis Mes        | Stop  |
| 004                                                                                                                    | 0      | 0      | G00 G98 G97 G40 |       |
| 005                                                                                                                    | 0      | 0      |                 |       |
| 006                                                                                                                    | 0      | 0      |                 |       |
| 007                                                                                                                    | 0      | 0      |                 |       |
| 008                                                                                                                    | 0      | 0      | MDI             |       |
| 009                                                                                                                    | 0      | 0      |                 |       |
| ADRS                                                                                                                   |        |        |                 |       |

Fig. 3-6-1-2

Function settings for the screw pitch compensation number, please refer to Machine Debugging-Screw Pitch Compensation.

### 3.6.2 Modification and Setting for Parameter and Screw Pitch Compensation Value

There are four methods for modifying and setting parameter and screw pitch compensation value.

1. Modify it in the page:
  - 1) Select <MDI> operation mode.
  - 2) Enter <Setting> page, and turn on the parameter switch in 【Setting】interface. 【Password】is required to be input for some parameter modification, See 3.5.1 for detailed operation.



- 3) Press  key to enter the parameter display page.
- 4) Move the cursor to the position where the parameter to be modified locates.

- (1). Press the page key to display the page where the parameter to be altered. Press the direction key (use left and right keys in bit parameter) to select the parameter to be altered.

- (2). Press  key and input the sequence number of the parameter to be modified. Then press



key to confirm it, and the cursor will move to the number directly.

- 5) Input new parameter value by numerical keys.

- 6) Press  key, the parameter value is input and displayed.

- 7) Turn off the parameter switch after parameter setting and confirmation is finished.

## 2. Loading and saving with backup disk

### 1) Loading parameter value from backup (see Fig.3-6-2-1)

The system has parameter and screw pitch compensation backup function. Take parameter value as an example bellow. The backup function of screw pitch compensation value is similar to that of the parameter value. Therefore, there is no further explanations. In parameter value I/O interface, select parameter input and output function by moving operation keys up and down. When parameter output function is selected, the parameter being used (including bit parameter and data parameter) can be backed up by the system. Back up files according to selected A, B, C OR D (selected by left and right operation keys). A and B are read-only disks, while C and D are read-write disks. Parameters in disk A and B are set in factory (A is default value for servo drive and B is default value for step drive). Disk A, B, C or D can be processed during loading, while only disk C, D can be processed during saving. When selecting parameter loading function, select A, B, C or D by left and right operation keys, namely, electronic disk number of the parameter file is selected. Then press input key to finish the loading, the parameter that CNC is using becomes the newly input parameter. Detailed operation process is as follows:

- ① Set the parameter switch as ON.
- ② Select <MDI> mode.

- ③ Press  key to enter parameter display interface

- ④ Press  key to enter parameter value I/O page.

- ⑤ Move the cursor up and down to “Parameter loading” position. Select the disk to be loaded by left and right key.

- ⑥ By pressing  key, the system will open corresponding electronic disk from the storage and load the data. After that, “Loading completion” is displayed in the lower left part of LCD.

### 2) Store the parameter to the disk (See the Fig. 3-6-2-1)

- ①～④ are the same as that above.

- ⑤ Move the cursor up and down to “Parameter storing” position. Select the disk to be loaded by left and right key.

- ⑥ By pressing  key, the system will write the parameter value into the user selected electronic disk. After that, “input completion” is displayed in the lower left part of LCD.

If “The electronic disk”, ”Parameter” and ”Backup” already exist in user partition, new parameter will cover the original parameter.

|                                                                                                  |             |                 |       |
|--------------------------------------------------------------------------------------------------|-------------|-----------------|-------|
|  ParameterValue | 00001 N0000 | Message         | 10:13 |
| _ReadParam: *A B C D                                                                             |             | Axis Rev        | 0000  |
| SaveParam: *C D                                                                                  |             | Tool No         | 0100  |
| <IN>read, <OUT>save                                                                              |             | Fact Rate       | 0     |
| A,B: default (A:servo, B:DY3)                                                                    |             | Cool Mes        | OFF   |
| C,D: user                                                                                        |             | Axis Mes        | Stop  |
|                                                                                                  |             | G00 G98 G97 G40 |       |
|                                                                                                  |             | MDI             |       |

Fig. 3-6-2-1

The disk loading and storage of the screw pitch error compensation parameter are similar to that of the system parameter. In screw pitch parameter interface, press  key to enter screw compensation electronic disk operation panel.

3. From CNC to CNC

The parameter and screw pitch error compensation can communicate with each by two different systems. For detailed operation, see section10.5 Communication between CNC and CNC.

4. Transmission with software

Connect PC and CNC with communication cable, turn on the CNC program switch, and open user communication software in PC. See the chapter ten Serial port communication for detailed operations.

3.7 Diagnosis Display

DI/DO signal state between CNC and the machine, system interface signal, keys state of MDI panel, CNC input/output state, system internal CNC state signal, etc. are displayed on the diagnosis interface. Please refer to appendix three for corresponding significances of each diagnosis number.

This diagnosis is used for detecting the operation state between CNC interface signal and internal signal, which can not be modified.

3.7.1 Diagnosis Data Display

Press  key to enter diagnosis interface, which consists of 【Diagnosis】 , 【Machine Panel】 , 【Macro Variable】 three display modes. They can be viewed and modified by pressing  key repeatedly.

1. 【Diagnosis】 interface can be viewed by pressing  or  key. See figure 3-7-1-1.

| DIAGNOSTIC                                                                          |          | 00001 N0000 | Message  | 10:09           |
|-------------------------------------------------------------------------------------|----------|-------------|----------|-----------------|
| No.                                                                                 | DATA     | No.         | DATA     |                 |
| 001                                                                                 | 00000000 | 009         | 00001100 | Axis Rev 0000   |
| 002                                                                                 | 01000001 | 010         | 00000000 | Tool No 0100    |
| 003                                                                                 | 01000100 | 011         | 00000001 | Fact Rate 0     |
| 004                                                                                 | 00100000 | 012         | 00000000 | Cool Mes OFF    |
| 005                                                                                 | 01000000 | 013         | 00000000 | Axis Mes Stop   |
| 006                                                                                 | 00000000 | 014         | 10001111 | G00 G98 G97 G40 |
| 007                                                                                 | 00000000 | 015         | 00000000 |                 |
| 008                                                                                 | 00000000 | 016         | 00000000 |                 |
| Machine tool side input signal(I0board)<br>Bit0:TCP DIQP DECX DITW *SP ST DECZ *ESP |          |             |          |                 |
| No.                                                                                 | 001      |             |          | MDI             |

Fig. 3-7-1-1

In diagnosis display interface, there are three diagnosis contents display columns in the lower part of the LCD. They display the contents of the diagnosis number where the cursor locates. See the *appendix III* for corresponding detailed significances of each diagnosis number.

2. See figure 3-7-1-2 for the contents of 【Machine Panel】 interface.

| OPERATOR             |       | 00001 N0000 | Message         | 10:13 |
|----------------------|-------|-------------|-----------------|-------|
| MLK (KEY1) : *OFF ON |       |             | Axis Rev        | 0000  |
| AFL (KEY2) : *OFF ON |       |             | Tool No         | 0100  |
| SBK (KEY3) : *OFF ON |       |             | Fact Rate       | 0     |
| DRN (KEY4) : *OFF ON |       |             | Cool Mes        | OFF   |
| BDT (KEY5) : *OFF ON |       |             | Axis Mes        | Stop  |
| POSITION (RELATIVE)  |       |             | G00 G98 G97 G40 |       |
| U                    | 0.000 |             |                 |       |
| W                    | 0.000 |             |                 |       |
|                      |       |             |                 | MDI   |

Fig. 3-7-1-2

When the machine panel displays pages, use "1. 2. 3. 4. 5" keys to switch Machine lock, M.S.T Lock,

Single Block, Dry run, Block skip. Their functions are the same as that of the



keys on the panel.



3. See figure 3-7-1-3 for the contents of 【Macro variable】 interface.

|                                                                                         |       |     |       |             |      |                 |       |
|-----------------------------------------------------------------------------------------|-------|-----|-------|-------------|------|-----------------|-------|
|  MACRO |       |     |       | 00001 N0000 |      | Message         | 10:09 |
| No.                                                                                     | DATA  | No. | DATA  | Axis Rev    | 0000 | G00 G98 G97 G40 |       |
| 200                                                                                     | 0.000 | 208 | 0.000 | Tool No     | 0100 |                 |       |
| 201                                                                                     | 0.000 | 209 | 0.000 | Fact Rate   | 0    |                 |       |
| 202                                                                                     | 0.000 | 210 | 0.000 | Cool Mes    | OFF  |                 |       |
| 203                                                                                     | 0.000 | 211 | 0.000 | Axis Mes    | Stop |                 |       |
| 204                                                                                     | 0.000 | 212 | 0.000 |             |      |                 |       |
| 205                                                                                     | 0.000 | 213 | 0.000 |             |      |                 |       |
| 206                                                                                     | 0.000 | 214 | 0.000 |             |      |                 |       |
| 207                                                                                     | 0.000 | 215 | 0.000 |             |      |                 |       |
| POSITION (RELATIVE)                                                                     |       |     |       |             |      |                 |       |
| U -169.380 W 122.337                                                                    |       |     |       |             |      |                 |       |
| No. 200                                                                                 |       |     |       | MDI         |      |                 |       |

Fig. 3-7-1-3

There are two parts in macro variable interface, which are #200～#231 power-off clearing and #500～#515 power-off memory. See variable instructions in **User Macro Variable** for detailed functions.

In macro variable interface, macro variable setting and modification are the same as that of the tool compensation. See variable instructions in **User Macro Variable** for detailed functions.

## Chapter Four Manual Opoeration



Press the  to enter manual operation mode, which mainly includes the manual feed, the spindle control and the machine panel control etc.

### 4.1 Coordinate Axis Movement

In the mode of the manual operation, X and Z axes can be operated at the manual feedrate or the manual rapid traverse rate.

#### 4.1.1 Manual Feed



Press the feed axis  or  key, and arrow indicates the direction here. The axis movement is stopped after releasing the key. In this case, the feed override can be adjusted to change the feedrate and the operation is similar to X-axis.

#### 4.1.2 Manual Rapid Traverse



Press  key and enter the manual rapid traverse state after the indicator  on the upper part of the edit panel is lighted up, and then press the key of feed axis, each axis moves at the rapid traverse rate.

**Note 1:** The feedrate performed in manual rapid traverse, the time constant and the acceleration/ deceleration modes are same as rapid traverse rate specified by program commands (G00 positioning).

**Note 2:** When rapid traverse is active (Rapid traverse indicator  lights up), NO.005 Bit6 determines whether the rapid move speed is manual feedrate or rapid traverse speed before the reference position returns after power on.

**Note3:** In Edit/MPG mode, the key is inactive and the indicator is off. Rapid feed can be selected in other modes and it is cancelled after the other mode is shifted.

#### 4.1.3 Manual Feed and Manual Rapid Traverse Rate Selection



When the manual feed operation is performed, federate override adjustment keys on the panel can be used to select the manual federate override (16 in total, 0%~150%).

See the table 4-1-3-1:

Table 4-1-3-1

| Feedrate override (%) | Feedrate (mm/min) |
|-----------------------|-------------------|
| 0                     | 0                 |
| 10                    | 1.5               |
| 20                    | 3.2               |
| 30                    | 5.0               |
| 40                    | 7.9               |
| 50                    | 12.6              |
| 60                    | 15                |
| 70                    | 32                |
| 80                    | 50                |
| 90                    | 79                |
| 100                   | 126               |
| 110                   | 200               |
| 120                   | 320               |
| 130                   | 500               |
| 140                   | 790               |
| 150                   | 1260              |

**Note:** The table has an error margin of 3%.

When the manual rapid traverse is performed, the manual rapid traverse rate can be selected by pressing

rapid traverse adjustment keys . Five gears rapid override are available: Fo, 25%, 50%, 75% and 100% (Fo is set by the data parameter P32).

**Note 1:** The rapid override selection is active for the following traverse speed

- (1) G00 rapid feed
- (2) Rapid feed in canned cycle
- (3) Rapid feed in the command G28
- (4) Manual rapid feed
- (5) Rapid feed in the manual reference return

**Example:** when the rapid feedrate is 8 m/min, and if the override is 50%, the speed is 4 m/min.

**Note 2:** The rapid override is inactive when the axis is moving.

## 4.2 Spindle Control

### 4.2.1 Spindle CW (Negative rotation)



: S rotation speed can be specified in MDI mode, the spindle rotates CW by pressing this key in the

mode of manual/MPG/single step.

#### 4.2.2 Spindle CCW (Positive rotation)



: S rotation speed can be specified in MDI mode, the spindle rotates CCW by pressing this key in the mode of manual/MPG/single step.

#### 4.2.3 Spindle Stop



: The spindle stops in the mode of manual/MPG/single step by pressing this key.

### 4.3 Other Manual Operations

#### 4.3.1 Cooling Control



: It is a compound key. The cooling is shifted between on and off. Its ON/OFF state can be viewed on LCD screen.

#### 4.3.2 Lubricating Control



: It is a compound key. The lubricating function is shifted between on and off.

#### 4.3.3 Manual Tool Change



: In Manual/MPG/Single step mode, by pressing this key, tool post rotates to change for another tool. Total tool number can be set by the data parameter P81. See the machine tool manufacturer's manual.

### 4.4 Tool Setting

Several tools are needed for processing a part. Because the tool installation error and tool error exist, tool nose positions do not entirely coincide when the tools rotate to the cutting position. In order to facilitate user programming without caring about the deviation between tools, the system is designed with tool offset automatic generating method. After tool setting, user makes programs according to part drawing and machining techniques without having to concern the deviation between tools. User only have to call corresponding tool compensation value in programs. The system is designed with many modes such as fixed-point tool setting, tool setting by test cutting and tool setting by machine zero return.

## 4.4.1 Fixed-point Tool Setting

When setting offset in  interface, if press  key after inputting the address key U (or W), the current relative coordinate value is set as offset value corresponding to this address.

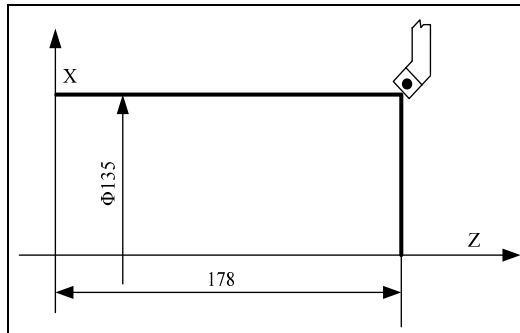


Fig. A

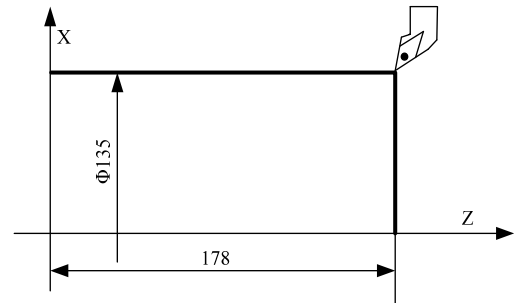


Fig. B

Fig. 4-4-1-1

Establish a workpiece coordinate system before the fixed-point tool setting.

Steps:

- 1) Select a tool without tool offset at random (usually the 1<sup>st</sup> tool, this tool will be used as the reference tool)
- 2) Position the tool nose of the reference tool to a point (tool setting point), as shown in Fig. A
- 3) Clear the relative coordinate (U, W). See section 3.1.1 for detailed operations

- 4) Press  key and move the cursor by  ,  keys to select a offset and take it as the offset of the reference tool.

First confirm whether the X and Z axial tool offset values are zero or not. If it is not zero, clear it as the following method.

Select number 001~064 tool offset by moving the cursor, input X0, Z0 to clear it.

- 5) After the tool is moved to a safe position, select another tool and move it to the setting point. As shown in Fig. B.

- 6) Press  key and select a offset in 001~064 as the offset of the tool. Then confirm whether the X and Z axial tool offset values are zero or not. If it is not zero, clear it as above step 4) introduced.

- 7) Press address key  , then press  key to input the tool offset value of X axis into the corresponding offset number of that.

- 8) Press address key  , hen press  key to input the tool offset value of Z axis into the corresponding offset number of that.

- 9) Repeat the steps from 5 to 8 to set the tool offset for other tools.

#### 4.4.2 Tool Setting by Test Cutting

When the workpiece coordinate system is not changed, perform tool setting by test cutting. Operation steps are as follows (Establish coordinate system on the part end surface):

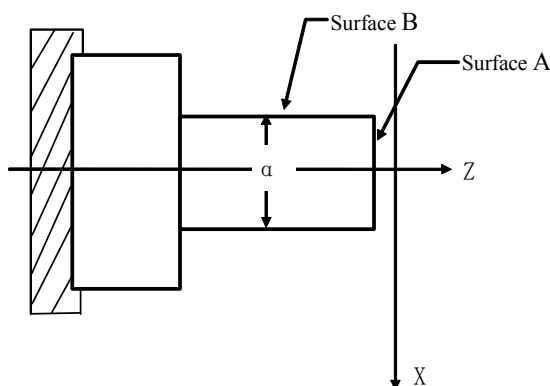


Fig. 4-4-2-1

1. Select a tool without tool offset at random (supposing it is the 1<sup>st</sup> tool)
2. In manual mode, it cuts along surface A in negative direction of X axis.
3. Retract the tool along the positive direction of the X axis when Z axis is not moving and stop the spindle.
4. Press **OFFSET** key to enter the Offset interface, and move the cursor by  and  keys to select offset 101 and take it as the offset of the tool.
5. Key in by sequence of the address key "Z", numerical key "0" and **DATA INPUT** key.
6. In manual mode, it cuts along surface B in negative direction of Z axis.
7. Retract the tool along the positive direction of the Z axis when X axis is not moving and stop the spindle.
8. Measure diameter "a" (Supposing  $a'=30$ )
9. Press **OFFSET** key to enter the Offset interface, and move the cursor by  and  keys to select offset 101.
10. Key in by sequence of the address key "X", numerical key "3", "0" and **DATA INPUT** key.
11. Move the tool to the safe position for tool changing.
12. Change another tool (supposing it is the 2<sup>nd</sup> tool) without tool offset.

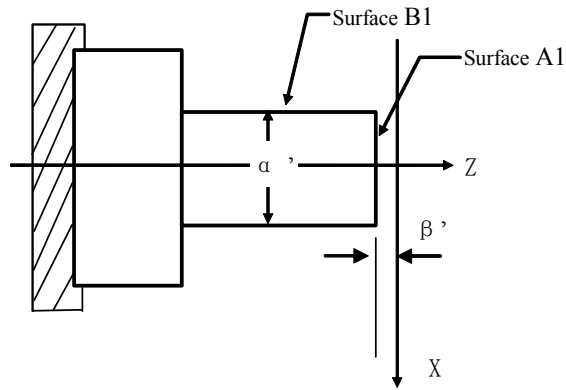


Fig. 4-4-2-2

13. In manual mode, it cuts along surface A1 in negative direction of X axis.
14. Retract the tool along the positive direction of the X axis when Z axis is not moving and stop the spindle.
15. Measure the distance "β" from surface A1 to the workpiece coordinate system (Supposing β=1).

16. Press  key to enter the Offset interface, and move the cursor by  ,  keys to select offset of 102.

17. Key in by sequence of the address key "Z", symbol key "-", numerical key "1" and  key.

18. In manual mode, it cuts along surface B1 in negative direction of Z axis.
19. Retract the tool along the positive direction of the Z axis when X axis is not moving and stop the spindle.
20. Measure the diameter "α'" (Supposing α'=28)

21. Press  key to enter the Offset interface, and move the cursor by  ,  keys to select offset of 102.

22. Key in by sequence of the address key "X", numerical key "2", "8" and  key.

23. Repeat the steps from 11 to 22 to perform tool setting operation for other tools.

#### 4.4.3 Tool Setting by Machine Zero Return

There is no reference tool in this tool setting method. When the tool is worn or to be adjusted, it only needs to be set again. Machine zero return should be done before the tool setting (See section 8.2 in this manual for machine zero return). The machining can be continued by performing machine zero return once after reset, which is convenient for the operation.

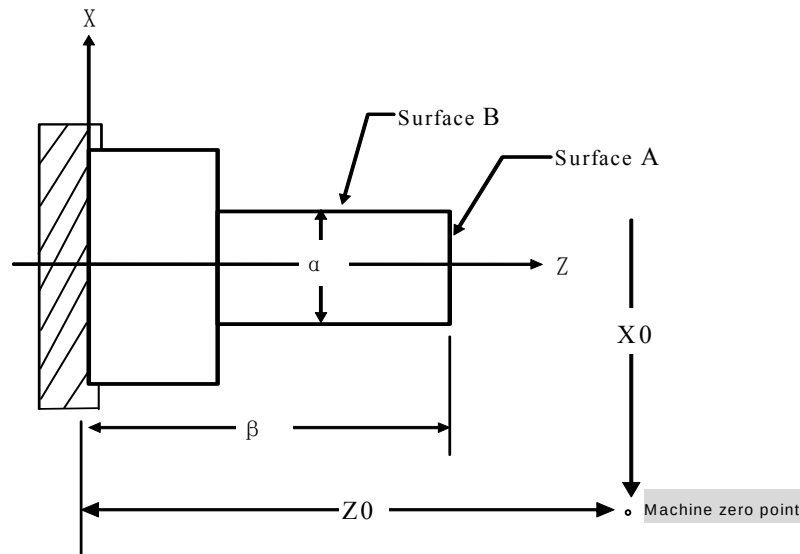


Fig.4-4-3-1

Operation steps are as follows:



- 1) Press  key to enter Machine Zero Return mode. Press positive direction keys of the X and Z axes to make the two axes return to machine zero point.

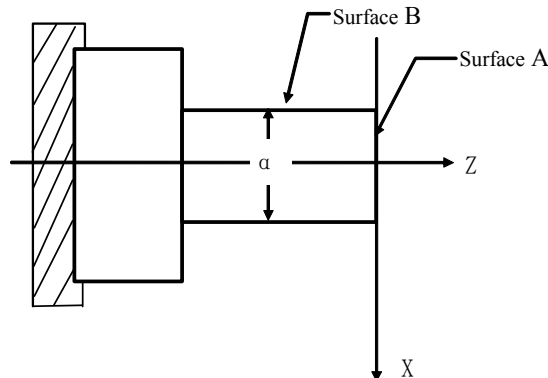


Fig. 4-4-3-2

- 2) Select a tool without tool offset at random (supposing it is the 1<sup>st</sup> tool)
- 3) In manual mode, it cuts along surface A in negative direction of X axis.
- 4) Retract the tool along the positive direction of the X axis when Z axis is not moving and stop the spindle.

- 5) Press  key to enter the Offset interface, and move the cursor by  ,  keys to select offset 101 as the offset number of the tool.

- 6) Key in by sequence of the address key "Z", numerical key "0" and  key.
- 7) In manual mode, it cuts along surface B in negative direction of Z axis.
- 8) Retract the tool along the positive direction of the X axis when Z axis is not moving and stop the spindle.
- 9) Measure the diameter "α'" (Supposing  $\alpha'=30$ )

10) Press  key to enter the Offset interface, and move the cursor by ,  keys to select offset of 101.

11) Key in by sequence of the address key “X”, numerical key “3”, “0” and  key.

12) Move the tool to the safe position for tool changing.

13) Change another tool (supposing it is the 2<sup>nd</sup> tool) without tool offset.

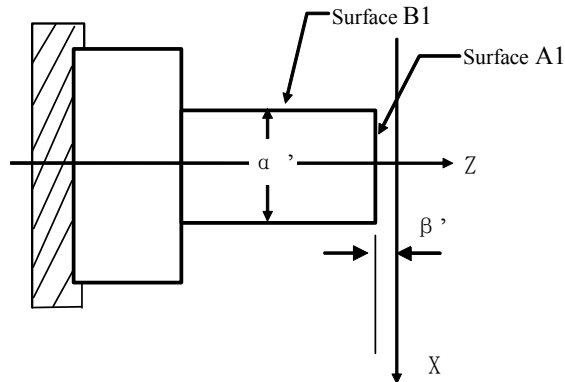


Fig. 4-4-3-3

14) In manual mode, it cuts along surface A1 in negative direction of X axis.

15) Retract the tool along the positive direction of the X axis when Z axis is not moving and stop the spindle.

16) Measure the distance “ $\beta$ ” from surface A1 to the workpiece coordinate system (Supposing  $\beta=1$ ).

17) Press  key to enter the Offset interface, and move the cursor by ,  keys to select offset of number 102.

18) Key in by sequence of the address key “Z”, symbol key “-”, numerical key “1” and  key.

19) In manual mode, it cuts along surface B1 in negative direction of Z axis.

20) Retract the tool along the positive direction of the Z axis when X axis is not moving and stop the spindle.

21) Measure diameter “a” (Supposing  $\alpha'=28$ )

22) Press  key to enter the Offset interface, and move the cursor by ,  keys to select offset of number 102.

23) Key in by sequence of the address key “X”, numerical key “2”, “8” and  key.

24) Repeat the steps from 12 to 23 to perform tool setting operation for other tools.

## 4.5 Offset Alteration

Offset can be input by U, W.

Example: If 0.010mm is need to be added to the offset value of the Z axis, press  key to enter the

Offset interface, and move the cursor by ,  keys to select a offset number of 001, then

input U0.010 and press  key, and 0.010mm is added to the original offset value automatically by the system.

## Chapter Five MPG/Sing Step Operation



Press key to enter MPG/ Single step mode, which can be selected by bit parameter NO:1#3. In the MPG/single step feed mode, the machine moves according to the defined step length in the system each time.

### 5.1 Step Feed

Set parameter **NO:1#3** to 0, and press to enter Step operation mode. The move increment can be selected by pressing any key of , , , and it is displayed on the interface. The step increment 0.100 is displayed on <Position> interface by pressing key, (See figure 5-1-1) :

| POSITION (RELATIVE) |                   | Message         | 09:03 |
|---------------------|-------------------|-----------------|-------|
| <b>O0001</b>        | <b>N0000</b>      | Axis Rev        | 0000  |
| <b>U</b>            | <b>0.000</b>      | Tool No         | 0100  |
| <b>W</b>            | <b>0.000</b>      | Fact Rate       | 0     |
|                     |                   | Cool Mes        | OFF   |
|                     |                   | Axis Mes        | Stop  |
|                     |                   | G00 G98 G97 G40 |       |
| STEP INC 0.100      | SPDLRIDE 100%     | STEP            |       |
| RAPDRIDE 100%       | WORK NO. 0        |                 |       |
| OVERRIDE 100%       | RUN TIME 00:00:00 |                 |       |

Fig. 5-1-1

Press X- or X+ key, it can move the X axis negatively or positively by a step increment. Press Z- or Z+ key, it can move the Z axis negatively or positively by a step increment.

**Note 1:** The speed of single step movement and that of the manual feed are the same.

**Note 2:** Rapid override and federate can be altered in Single Step mode.

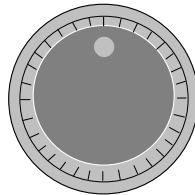
### 5.2 MPG (Manual pulse generator) Feed

Set parameter **NO:1#3** to 1, and press to enter MPG operation mode. The move increment can be selected by pressing any key of , , , and it is displayed on the interface. The MPG increment 0.100 is displayed on <Position> interface by pressing key, (See figure 5-2-1) :

| POSITION (RELATIVE) |                   | Message         | 09:03 |
|---------------------|-------------------|-----------------|-------|
| <b>O0001</b>        | <b>N0000</b>      | Axis Rev        | 0000  |
| <b>U</b>            | <b>0.000</b>      | Tool No         | 0100  |
| <b>W</b>            | <b>0.000</b>      | Fact Rate       | 0     |
|                     |                   | Cool Mes        | OFF   |
|                     |                   | Axis Mes        | Stop  |
|                     |                   | G00 G98 G97 G40 |       |
| HNDL INC 0.100      | SPDLRIDE 100%     | HANDLE (X)      |       |
| RAPDRIDE 100%       | WORK NO. 0        |                 |       |
| OVERRIDE 100%       | RUN TIME 00:00:00 |                 |       |

Fig. 5-2-1

THE MPG EXPERIENCE IS AS THE FIGURE BELLOW:



Manual pulse generator (MPG)

### 5.2.1 Moving Axis and Direction Selection

In MPG mode, press ,  or  key, the corresponding axis is selected and it will be controlled by MPG. For example, X axis can be moved by moving MPG after  key is pressed (See figure 5-2-1-1).

Generally, MPG CW is for positive feed, and CCW for negative feed.

Blinking →

| POSITION (RELATIVE) |                   | Message         | 09:03 |
|---------------------|-------------------|-----------------|-------|
| <b>O0001</b>        | <b>N0000</b>      | Axis Rev        | 0000  |
| <b>U</b>            | <b>0.000</b>      | Tool No         | 0100  |
| <b>W</b>            | <b>0.000</b>      | Fact Rate       | 0     |
|                     |                   | Cool Mes        | OFF   |
|                     |                   | Axis Mes        | Stop  |
|                     |                   | G00 G98 G97 G40 |       |
| HNDL INC 0.100      | SPDLRIDE 100%     | HANDLE (X)      |       |
| RAPDRIDE 100%       | WORK NO. 0        |                 |       |
| OVERRIDE 100%       | RUN TIME 00:00:00 |                 |       |

Fig. 5-2-1-1

The MPG (Manual pulse generator) feed direction is defined by its rotation direction. See the manufacturer's instructions for details. Generally, MPG CW is for positive feed and CCW is for negative feed.

### 5.2.2 Instructions for MPG Feed

1. Relationship between MPG scale and machine movement amount is shown in the table below:

Table 5-2-2-1

|                                 | Movement amount of each MPG scale |      |     |
|---------------------------------|-----------------------------------|------|-----|
| MPG increment<br>(mm)           | 0.001                             | 0.01 | 0.1 |
| Machine movement<br>amount (mm) | 0.001                             | 0.01 | 0.1 |

2. Values in the above table vary with mechanical transmission. See manufacturer's instructions for details.
3. The rotation speed of the MPG should be less than 5 r/s. If it is over 5 r/s, the scale may not coincide with the movement amount.

4. The  and  keys are only active in MPG/Step mode.

### 5.3 The Miscellaneous Control in MPG/Step Operation

It is the same as the manual operation mode. Refer to the section 4.2 and 4.3 of this operation manual for details.

## Chapter Six MDI Operation

The MDI operation function is added besides data input, parameter and offset alteration functions. Commands that are input directly can be executed with this function. There are detailed instructions for MDI, parameter and offset alteration in *interface display and data alteration and setting*, chapter 3. MDI operation function is introduced in this chapter.

### 6.1 MDI Blocks Input

Program input in **【MDI】** mode is the same as that of the Edit mode, *see edit operations*, chapter 7. Input steps in **【MDI】** mode are introduced bellow.

Example: Steps for inputting a block G00 X50 Z50 by **【MDI】** operation interface are as follows:

1. Press  key to enter MDI mode

2. Press  key to enter **【MDI】** operation interface, see section 3.2 for detailed operations. (See figure 6-1-1)

3. After the address key G00 is input, press  key to confirm the operation and G00 is displayed on LCD screen. Input X50 and press  key to confirm. Then input Z50 and press  key to confirm. At this time, programs are input and can be viewed in the interface, see the figure bellow:

|                                                                                     |          |               |                 |       |
|-------------------------------------------------------------------------------------|----------|---------------|-----------------|-------|
|  | PROGRAM  | 00001 N0000   | Message         | 10:04 |
| G00                                                                                 | X 50.000 |               | Axis Rev        | 0000  |
|                                                                                     | Z 50.000 | G94 F 250.000 | Tool No         | 0100  |
|                                                                                     | U        | G97 M         | Fact Rate       | 0     |
|                                                                                     | W        | G98 S 0       | Cool Mes        | OFF   |
|                                                                                     | R        | T 0100        | Axis Mes        | Stop  |
|                                                                                     | F        |               | G00 G98 G97 G40 |       |
|                                                                                     | M        | G21           |                 |       |
|                                                                                     | S        | G40           |                 |       |
|                                                                                     | T        |               |                 |       |
|                                                                                     | P        |               |                 |       |
|                                                                                     | Q        | SACT 0000     |                 |       |
| ADRS                                                                                |          |               | MDI             |       |

Fig. 6-1-1

### 6.2 MDI Block Operation and Stop

The MDI can be performed by pressing  key after the command is input as the steps in section

6.1. The block operation can be stopped by pressing  or pressing  key during operation.

**Note:** MDI operation must be performed in the MDI mode!

### 6.3 MDI Block Words Alteration and Clearing

If there are errors during words input, press  key to cancel it word by word. If the errors are found after it is finished, input right one to take the place of the wrong one or press  key to clear all contents, and then input it again.

### 6.4 Conversion of Operation Modes

The MDI mode operation may stop immediately after shifting to the other modes. Specific as follows:

- 1) After shifting to Edit, AUTO mode, the machine is stopped after the current block is performed.
- 2) After shifting to Manual, MPG, Step mode, the machine is stopped immediately.
- 3) After shifting to Machine zero return interface, the machine decelerates to stop.

## Chapter Seven Edit Operation

### 7.1 Program Edit

To prevent the program from being modified or deleted by people at random, the system is designed with a program switch. The switch must be turned on before program editing. Please refer to switch setting in section 3.5.1.

Workpiece program editing should be performed in edit mode. Press  key to enter edit mode.

Press  key to enter program display interface, and the program can be edited and altered (See Fig. 7-1-1).

|                                                                                           |             |                 |       |
|-------------------------------------------------------------------------------------------|-------------|-----------------|-------|
|  PROGRAM | 00001 N0000 | Message         | 09:04 |
| 00001;                                                                                    |             | Axis Rev        | 0000  |
| G50 X250. Z450. ;                                                                         |             | Tool No         | 0100  |
| T0101;                                                                                    |             | Fact Rate       | 0     |
| G00 X100. Z200. ;                                                                         |             | Cool Mes        | OFF   |
| G90 U-10. W-50. R-1.5 F500. ;                                                             |             | Axis Mes        | Stop  |
| G90 U-50. W-10. R-3. F350. ;                                                              |             | G00 G98 G97 G40 |       |
| G00 X120. ;                                                                               |             |                 |       |
| T0202;                                                                                    |             |                 |       |
| M03 S01;                                                                                  |             |                 |       |
| G92 X80. W-65. R5.1 F0.5;                                                                 |             |                 |       |
| ADRS                                                                                      | Row 2       | EDIT            |       |

Fig. 7-1-1

#### 7.1.1 Program Creation

##### 7.1.1.1 Creation of the Program Number

By setting the parameter NO:2#7, it is able to set whether the block number is active or not. If the parameter is set to 1, the block number is inserted automatically during program editing, increment of the block number is set by the data parameter P50.

##### 7.1.1.2 Input and Storage of the Program

1. Press  key to enter the Edit mode.

2. Press  key to enter the Program interface (See figure 7-1-1-2-1)

|                                                                                                                                                                                               |             |                 |       |
|-----------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|-------------|-----------------|-------|
|  PROGRAM                                                                                                     | 00001 N0000 | Message         | 09:04 |
| 00001;<br>G50 X250. Z450. ;<br>T0101;<br>G00 X100. Z200. ;<br>G90 U-10. W-50. R-1.5 F500. ;<br>G90 U-50. W-10. R-3. F350. ;<br>G00 X120. ;<br>T0202;<br>M03 S01;<br>G92 X80. W-65. R5.1 F0.5; |             | Axis Rev        | 0000  |
|                                                                                                                                                                                               |             | Tool No         | 0100  |
|                                                                                                                                                                                               |             | Fact Rate       | 0     |
|                                                                                                                                                                                               |             | Cool Mes        | OFF   |
|                                                                                                                                                                                               |             | Axis Mes        | Stop  |
|                                                                                                                                                                                               |             | G00 G98 G97 G40 |       |
| ADRS                                                                                                                                                                                          | Row 2       | EDIT            |       |

Fig. 7-1-1-2-1

3. Press address key , and key in numerical keys by sequence of , , ,  (Program O3333 creation for an example). O3333 is displayed behind the numerical column. See the figure bellow (Fig. 7-1-1-2-2).

|                                                                                                                                                                                               |              |                 |       |
|-----------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|--------------|-----------------|-------|
|  PROGRAM                                                                                                    | 00001 N0000  | Message         | 09:04 |
| 00001;<br>G50 X250. Z450. ;<br>T0101;<br>G00 X100. Z200. ;<br>G90 U-10. W-50. R-1.5 F500. ;<br>G90 U-50. W-10. R-3. F350. ;<br>G00 X120. ;<br>T0202;<br>M03 S01;<br>G92 X80. W-65. R5.1 F0.5; |              | Axis Rev        | 0000  |
|                                                                                                                                                                                               |              | Tool No         | 0100  |
|                                                                                                                                                                                               |              | Fact Rate       | 0     |
|                                                                                                                                                                                               |              | Cool Mes        | OFF   |
|                                                                                                                                                                                               |              | Axis Mes        | Stop  |
|                                                                                                                                                                                               |              | G00 G98 G97 G40 |       |
| ADRS                                                                                                                                                                                          | O 3333 Row 2 | EDIT            |       |

Fig. 7-1-1-2-2

4. Press  key to create a new program, as shown in the following figure (Fig. 7-1-1-2-3).

|                                                                                             |             |                 |       |
|---------------------------------------------------------------------------------------------|-------------|-----------------|-------|
|  PROGRAM | 03333 N0010 | Message         | 09:04 |
| 03333;<br>N10<br>%                                                                          |             | Axis Rev        | 0000  |
|                                                                                             |             | Tool No         | 0100  |
|                                                                                             |             | Fact Rate       | 0     |
|                                                                                             |             | Cool Mes        | OFF   |
|                                                                                             |             | Axis Mes        | Stop  |
|                                                                                             |             | G00 G98 G97 G40 |       |
| ADRS                                                                                        | Row 2       | EDIT            |       |

Fig. 7-1-1-2-3

5. Input the edited program word by word, and press  key after a block is finished.

Switch to other interface (e.g.  interface) after a program is finished, or switch to other working mode and at the same time, the program is saved automatically.

**Note 1: In Edit mode, only a complete word can be input. Single character or number can not be input.**

**Note 2: If there are errors during block inputting, press  key to cancel the whole block.**

#### 7.1.1.3 Search of the Sequence Number, Character and Line Number

Sequence number retrieval, which means searching a sequence number of a program, is used to start program execution or edit from the retrieved sequence number. The block skipped for searching has no effect on the state of the CNC. (The coordinate value, M, S, T code and G code of the skipped block have no effect on the coordinate value and mode value of the CNC).

If execution starts from a block of the retrieved program, the machine state and CNC state should be checked. In addition, they should be consistent with corresponding M, S, T code and the coordinate system setting (it can be set by MDI mode) before execution.

Character search is used to search the specified address or numerical characters in program, which is mainly used in edit program.

Steps of searching sequence number, characters in program:

1) Press  key to enter Edit mode.

2) Open the target program in  interface. See 6.1 for detailed operations.

There are two methods for searching characters:

1) Scanning: find the page where the target locates by the page key, and then move the cursor to the target position.

2) Search method: input the character or sequence number to be searched, and press  or  key to search it upward or downward. The same one downward can be searched, the alarm occurs if it is not searched.

#### 7.1.1.4 Cursor Positioning

In edit mode, select  key to enter program page.

- 1) Press  key to move up the cursor to the last line. When the line which the cursor is in is bigger than the end row of the last line, the cursor can be moved up to the end of the last line.
- 2) Press  key to move down the cursor to the next line. When the line which the cursor is in is bigger than the end row of the last line, the cursor can be moved down to the end of the next line longer
- 3) Press  to move right the cursor to one row. When the cursor is in the end of the line, it can be moved to the home of the next line.
- 4) Press  to move left the cursor to one row. When the cursor is in the home of the line, it can be moved to the end of the last line
- 5) Press  to page up the screen and the cursor moves to the same position of the last screen.
- 6) Press  to page down the screen and the cursor moves to the first row of the first line in the next screen.
- 7) Press  key, the cursor moves to the head of the program.

#### 7.1.1.5 Character Insertion, Deletion and Alteration

By the methods in 7.1.1.3 and 7.1.1.4, move the cursor to the position to be altered.

##### 1. Insertion of the character

After inputting the data, press  key, the input contents will be inserted at the right side of the cursor.

##### 2. Deletion of a character

Move the cursor to the character to be deleted, and the character where cursor locates will

be deleted by pressing  key. If the  key is pressed continuously, the contents at the right side of the cursor will be deleted one by one.

##### 3. Alteration of the character

Move the cursor to the character to be altered, then input the contents and press  key, the contents is input to take the place of the character where the cursor locates.

### 7.1.2 Deletion of a program

Steps for deleting a program in the memory are as follows:

- 1) Select <Edit> mode.

- 2) Press  key to enter the program display interface.

- 3) Press address key .

- 4) Input the name of the program to be deleted

- 5) Press  key, and the corresponding program in the memory will be deleted.

**Note:** If the program does not exist, system alarms (P/S alarm: 014 the program does not exist)

### 7.1.3 Deletion of All Programs

Steps for deleting all programs in the memory are as follows:

- 1) Select <Edit> mode.

- 2) Press  key to enter the program display interface.

- 3) Press address key .

- 4) Input -9999 and press  key, all the programs will be deleted.

### 7.1.4 Duplication of the Program

Copy the current program and save it with a new program name.

- 1) Select <Edit> mode.
- 2) Enter the program display interface, and open the program to be copied.

- 3) Press address key  and input new program number.

- 4) Press  key to complete the file duplication and enter a new program edit interface.

### 7.1.5 Rename of the Program

Change the current name to a new one:

- 1) Select <Edit> mode.
- 2) Enter program display interface.

- 3) Key in address  and input new program name.
- 4) Press  key to complete the performance.

**Note 1:** In program creation, if the input program name exists, this program will be opened.

**Note 2:** In program duplication or rename program, the name of the target program should not be the same as any one existed in the memory. Otherwise, the alarm occurs.

7.2 Program Management

7.2.1 Search of Program List

Press  key to enter the Program List page (See 7-2-1-1):

|                                                                                           |             |                 |       |
|-------------------------------------------------------------------------------------------|-------------|-----------------|-------|
|  PROGRAM | 00001 N0000 | Message         | 10:04 |
| SYSTEM EDIT:GSK-980TB3 (08) V3. 74. 083                                                   |             | Axis Rev        | 0000  |
| PRG. NO. USED: 13 FREE:687                                                                |             | Tool No         | 0100  |
| MEMORY USED: 704 FREE: 23840                                                              |             | Fact Rate       | 0     |
| PRG. LIBRARY:                                                                             |             | Cool Mes        | OFF   |
| 00000 00001 00002 00003 00004 00005                                                       |             | Axis Mes        | Stop  |
| 00006 00007 00008 00009 00010 00011                                                       |             | G00 G98 G97 G40 |       |
| 00045                                                                                     |             |                 |       |
|                                                                                           |             |                 |       |
| ADRS                                                                                      |             | MDI             |       |

Fig. 7-2-1-1

In this page, it displays all programs names saved in the memory in the form of program list. If one page can not display all programs, press  key to the next page.

7.2.2 Quantity of the Saved Programs

The memory of this system can save 500 programs. For quantity of saved programs, please refer to program list page in the above 7.2.1.

7.2.3 Memory Capacity

For detailed storage state, please refer to program list page in the above 7.2.1.

#### 7.2.4 Program Lock

To prevent the program to be altered or deleted by people at random, the system is designed with a program switch. It must be turned off after program editing. Please refer to section 3.5.1 for details.

### 7.3 Other Operations Available in Edit Mode

In the Edit mode, the following operations are available:

1. Spindle override adjustment.
2. Rapid override adjustment.

## Chapter Eight AUTO Operation

### 8.1 Selection of the Program To Be Run

#### 1. Program loading in Edit or Auto mode

- (a) Select the Edit or Auto mode

- (b) Press  key to enter the PRG display page

- (c) Press the address key  to input the name of the program to be performed and then press

-  key to search the program (Or after pressing  key once, press  or  key directly to search the program existed)

- (d) If the program name is searched, the program searched will be shown on the LCD screen. If the program does not exist, the alarm will be issued.

#### 2. Load programs in MDI mode

- (a) In MDI mode.

- (b) Press  key to enter the PRG display page, and move the cursor to the program to be performed.

- (c) Press  key to confirm.

- (d) Select the Edit or Auto mode, and press  key to enter the PRG display page, then the selected contents can be viewed on the LCD screen.

### 8.2 Start of the Auto Run

After the programs are selected by the two methods introduced in 8.1. In <AUTO> mode, press



key to start the program. <Position>, <Program> and <Graph> interfaces can be shifted to observe the program operation state.

Since the program execution begins from the block where the cursor locates,



before pressing the key, make a check whether the cursor is located at the block to be executed and whether each mode is correct. If begins from the start line, move the cursor to the start line by pressing



key in Edit mode, and press



key to execute the program from the beginning .

**Note:** If the cycle start key is pressed continuously, the system issues alarm that ERR191 overtime. The parameter P159 can be altered to adjust the key alarm time.

### 8.3 Auto Run Stop

In Auto run, there are five ways to stop the programs:



1. Stop by key



In Auto run, by pressing key or external dwell key, the machine remains at the following state:

- 1) The machine feed decelerate to stop
- 2) During the execution of the dwell command (G04), it pauses after G04 command execution is finished.
- 3) The modal function and state are saved



4) The program execution continues after pressing the key



2. Stop by Reset key

Please refer to section 2.3.1 operation manual.

3. Stop by code (M00)

After the block containing M00 is executed, the auto run is stopped. So the modal function and state



are all reserved. Press the key, the program execution continues

4. Stop by Emergency stop button

Please refer to section 2.3.2 operation manual.

5. Stop by mode switching

When the programs are being performed in Auto, MDI interface of MDI mode, the program can be stopped by switching to other modes. See the following for details:

- 1) When the Auto mode is switched to Edit, MDI interface, the machine can be stopped after the current block is executed.
- 2) When the Auto mode is switched to Manual, MPG, Step interface, the machine is stopped immediately.
- 3) When the Auto mode is switched to the Machine zero, the machine decelerates to stop.

### 8.4 Auto Run From an Arbitrary Block

The system supports auto run from an arbitrary block. Detailed operation steps are as follows:

1. Press  key to enter the Manual mode, and start the spindle and other miscellaneous functions.
2. Press  key to enter the Edit mode, open the programs to be executed. (See section 8.1 for detailed operations)
3. Move the cursor to the block to be executed;
4. Press  key to enter the Auto mode;
5. Press  key to start the program.

Note: When a block is selected, confirm the relationship between the selected block and the current absolute coordinate. If the program of tool change will be performed, ensure collision will not happen at the current position to avoid machine tool or people damage.

## 8.5 Dry Run

Before the program execution, in order to avoid the programming errors, the Dry run mode is selected to check the program. It is used with M.S.T lock and machine lock.

Method 1: Press  key to enter Auto mode, and press  (the indicator  on the upper part of the edit panel lights up, it means that the dry run function is selected).

Method 2: In Auto or MDI mode, press the  key continuously to enter the tool panel page, press numerical key 4 to select Dry run ON/OFF.

In Dry run mode, no matter how the federate is defined by the programs, the system runs at the rates in the following table:

Table 8-5-1

|                           | Program command                             |                                      |                 |
|---------------------------|---------------------------------------------|--------------------------------------|-----------------|
|                           | Rapid feed (see note 1)                     | Cutting feed (Except for screw feed) | Screw feed      |
| Rapid traverse switch ON  | Rapid feed                                  | Max. JOG feed                        | Screw lead *500 |
| Rapid traverse switch OFF | JOG federate or rapid feedrate (see note 2) | JOG feed                             |                 |

**Note 1:** X, Z axis speed of the rapid feed are set by the data parameters P21, P22 in the parameter page.

**Note 2:** By setting the parameter NO: 2#1, it is able to set whether the rapid traverse command in Dry run is active or not.

**Note 3:** If the dry run function is executed in Auto or MDI mode, it is still active when the Auto mode or MDI mode are switched to the other modes (Manual, MPG, Edit, Machine zero return, Program zero), but the ON/OFF of the Dry run can not be set at this time.

## 8.6 Single Block Execution

If the state of single block execution is to be checked, select Single Block execution.

Method 1: Press  key to enter Auto mode, and press  key (If the indicator  on the upper part of the edit panel lights up, it means that the Single Block execution is selected). In single mode, when the current block execution is finished, the CNC stops. If the next block is to be executed, it needs to press the  key, and then repeat this operation until the whole program is finished.

Method 2: In Auto or MDI mode, press the  key continuously to enter the Tool panel page, press the numerical key 3 to select Single block ON/OFF.

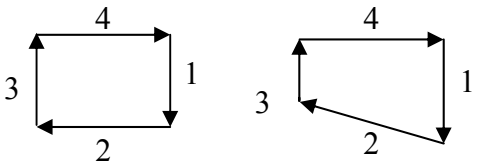
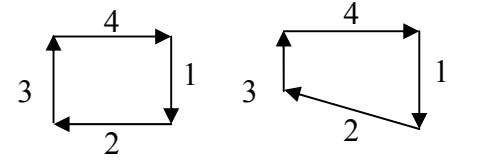
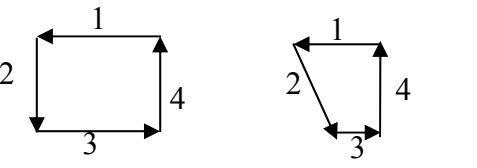
**Note 1:** Even at the mid point, the single block stops in G28.

**Note 2:** When the single block is on, and the canned cycle G90, G92, G94, G70~G75 are being executed, see the table 8-6 for details.

**Note 3:** Single block stop is available in programs M98\_; M99; and G65. By setting the system parameter P13 BIT0, it is able to set G65 single block execution is active.

**Note 4:** If the single block function is executed in Auto or MDI mode, it is still active when the Auto mode or MDI mode are switched to the other modes (Manual, MPG, Edit, Machine zero return, Program zero), but the ON/OFF of the Single block function can not be set at this time.

Table 8-6-1

| G code | Tool path                                                                           | Explanations                                                                  |
|--------|-------------------------------------------------------------------------------------|-------------------------------------------------------------------------------|
| G90    |  | Action 1 to 4 is a complete circulation. It stops after action 4 is finished. |
| G92    |  | Action 1 to 4 is a complete circulation. It stops after action 4 is finished. |
| G94    |  | Action 1 to 4 is a complete circulation. It stops after action 4 is finished. |

|          |  |                                                                                     |
|----------|--|-------------------------------------------------------------------------------------|
| G70      |  | Action 1 to 7 is a complete circulation. It stops after action 7 is finished.       |
| G71, G72 |  | 1~4, 5~8, 9~12, 13~16, 17~20 is a circulation, it stops after actions are finished. |
| G73      |  | 1 ~ 6 is a circulation, it stops after actions are finished.                        |
| G74, G75 |  | 1 ~ 10 is a circulation, it stops after actions are finished.                       |
| G76      |  | 1~4, 5~8, 9~12 is a circulation, it stops after actions are finished.               |

## 8.7 Machine Lock

Method 1: In Auto or MDI mode, press  key (the indicator  on the upper part of the edit panel lights up, it means that it has entered the machine lock state). Now each axis of the machine can not move, while the display of the coordinate is the same as that in machine operating, and M, S, T can be executed. This function is used for program check.



Method 2: In Auto or MDI mode, press key continuously to enter the Tool Panel page. Press the numerical key 1 to select ON/OFF of the machine lock.

**Note 1:** If the machine lock function is executed in Auto or MDI mode, it is still active when the Auto mode or MDI mode are switched to the other modes (Manual, MPG, Edit, Machine zero return, Program zero), but the ON/OFF of the machine lock can not be set at this time.

## 8.8 M. S. T Lock



Method 1: In Auto or MDI mode, press key (the indicator on the upper part of the edit panel lights up, it means that it has entered the M.S.T Lock state). Now the machine carriage moves without the M, S, T code being executed. The machine lock and M, S, T lock are used together to check the program.



Method 2: In Auto or MDI mode, press key continuously to enter the Tool Panel page. Press the numerical key 2 to select ON/OFF of the M.S.T lock.

**Note 1:** M00, M30, M98, M99 are executed according to the common practice.

**Note 2:** If the M.S.T Lock function is executed in Auto or MDI mode, it is still active when the Auto mode or MDI mode are switched to the other modes (Manual, MPG, Edit, Machine zero return, Program zero), but the ON/OFF of the M.S.T Lock can not be set at this time.

## 8.9 Block Skip



Method 1: In Auto or MDI mode, press key before execution (the indicator on the upper part of the edit panel lights up, it means that it has entered the Block Skip state). Now the program headed with “/” sign is skipped without execution.



Method 2: In Auto or MDI mode, press key continuously to enter the Tool Panel page. Press the numerical key 5 to select ON/OFF of the Block skip.

**Note 1:** If the block skip is executed in Auto or MDI mode, it is still active when the Auto mode or MDI mode are switched to the other modes (Manual, MPG, Edit, Machine zero return, Program zero), but the ON/OFF of the Block Skip can not be set at this time.

## 8.10 Adjustment of the Feedrate, Rapid Rate in Auto Mode

In Auto mode, the running speed can be altered by adjusting the feedrate override, rapid override.



In Auto mode, press keys on the panel to select the feedrate override, 16-level real-time feedrate adjustment can be obtained.



Press the key each time, the feedrate override ascends a level (10% for each level) till 150%



Press the key each time, the feedrate override descends a level (10% for each level) till 0.

**Note 1: The value specified by F is adjusted by feedrate override**

**Actual federate = value specified by F × feedrate override**

In Auto mode, press    keys on the panel to select the federate override, which includes Fo , 25% , 50% , 75% , 100% five gear levels. (Fo is set by the data parameter P32)

**Note 2: Following is calculation form of the rapid traverse rate obtained by data parameter P21, P22 setting and rapid override adjustment.**

**X axis actual rapid traverse rate = value set by parameter P21\*rapid override**

**When the rapid override is Fo, the actual rapid traverse rate is set by the data parameter P32 (Used for whole axis)**

**Calculation method for the actual rapid traverse rate of the Z axis is the same as above.**

### 8.11 Spindle Speed Adjustment in Auto Mode

While the spindle speed is controlled by the analog voltage output in Auto mode, it can be adjusted by spindle override

In Auto mode, press    key to adjust the spindle override for spindle speed, it can realize 8-level real-time override adjustment between 50% ~ 120%.

Press the  key each time, the spindle override ascends a level (10% for each level) till 120%.

Press the  key each time, the spindle override descends a level (10% for each level) till 50%.

The actual speed of the spindle = spindle speed of program command × spindle override

### 8.12 Cooling Control

Cooling ON/OFF can be shifted by pressing  key on the panel, and its state can be viewed on the LCD screen.

## Chapter Nine Zero Return Operation

### 9.1 Concept of the Machine Zero Return

The **machine coordinate system** is an inherent coordinate system of the machine. The origin of the machine coordinate system is called **machine zero** (or mechanical reference point). Usually it is fixed on the positive max. Strokes of X, Z axes. The machine zero is a fixed point, which is determined after the machine is designed, manufactured, and debugged. Normally, the machine zero point can not be recognized till the CNC device is turned on, the machine zero point return is performed in Auto or by manual operation.

By setting Bit0, 1 of the parameter NO:6, it is able to set whether a machine zero point on X, Z axis according to whether there is a zero return switch fixed on the machine.

- a) When the parameter NO:6 Bit0=0, Bit1=0, it is type A of zero return. The process of the reference return is as follows:

The zero return axis moves to the machine coordinate zero point (that is X0 or Z0) at the speed of G0. After reaching the zero point, the coordinate of the zero return axis is cleared, and machine zero return is finished.

- b) When the parameter NO:6 **Bit0=1, Bit1=1, Bit4=0, Bit5=0**, it is **type B** of zero return. The process of the reference return is as follows:

- ① The zero return coordinate axis moves along with its positive direction at the rapid traverse speed. As the axis presses down the deceleration switch, the coordinate axis decelerates to the minimum speed set by the system [the speed specified by the parameter **P043** and it moves reversely at this speed until the zero block departs from the deceleration switch.
- ② The zero return axis moves at the minimum speed [the speed is specified by the parameter **P043** until the one-turn signal (or the signal of approaching switch zero point) of the motor encoder is detected, it continues to move a short distance forward then travels reversely at the speed specified by the system. When the one-turn signal of the motor encoder is detected again, it reaches the zero point of the coordinate axis to stop. The machine coordinate of the zero return axis is cleared and the zero return completed.

Only one block is needed as a deceleration signal in type B, zero signal is PC signal provided by drive unit.

- c) When the parameter NO:6 **Bit0=1, Bit1=1, Bit4=1, Bit5=1**, it is **type C** of zero return. The process of the reference return is as follows:

- ① The zero return coordinate moves along with its positive direction at the rapid traverse speed. As the axis presses down the deceleration switch, the coordinate axis decelerates to the minimum speed set by the system [the speed specified by the parameter **P043** and it moves reversely at this speed.
- ② The system stops when the axis departs from the block, and this point is referred as the zero point. Then the machine coordinate of the zero return axis is cleared and the zero return completed. Only one block is needed as a deceleration and zero signal in type C.

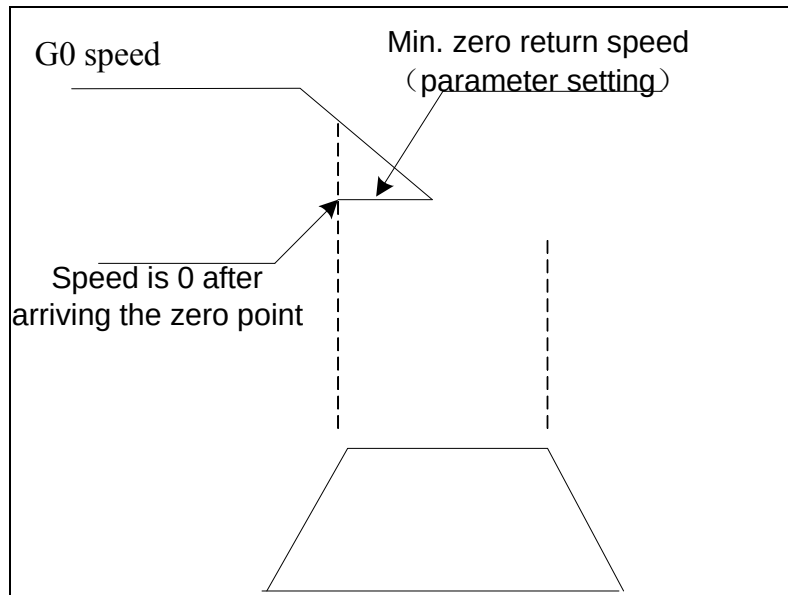


Fig.8-1-1

## 9.2 The Operation Procedures of Machine Zero Return

### 1. The procedures of type B, C



1. Enter the machine zero return mode by pressing , the “Machine Zero” is displayed on the LCD screen at this time.
2. The X, Y axis, which is to be returned the machine zero point, is selected
3. The machine moves along with its zero point direction, and it rapidly moves before decelerating. The machine moves to its zero point (it is also called as reference position) at the FL speed [The move speed is set by data parameter **P43** after the deceleration switch is pressed. The coordinate axis stops and zero return indicator lights up when the machine zero point is returned.

**Note 1:** The corresponding indicator lights up when the machine zero point return is completed.

**Note 2:** The indicator is power-off if operator moves out a corresponding axis from the machine zero point or press down the emergency button.

**Note 3:** Refer to the manual issued by the manufacturer for the machine zero point direction, then confirm it by Bit0 and Bit1 of the parameter N0:5. Before reference return, the tool post should be stopped in the negative position of the machine reference point, machine zero point (namely, reference point) direction. Please refer to the manual provided by the machine tool builder.

**Note 4:** Make sure Bit0 and Bit1 of the parameter N0:6 are set to 0 before the machine reference switch is installed on the machine. Select machine zero return type A, but do not perform zero return. Otherwise, accidents will occur because the tool post moves rapidly for there is no deceleration switch.

### 2. The procedures of type A

1. Press  key, press  or  key to enter **【Integrated】** position page. By pressing “X” key, X axis coordinates on the screen will be highlighted, and then press the  key. Machine zero return completes and indicator  lights up.

**Note:** Zero return operations of the Z axis are the same as above.

### 9.3 Concept of Program Zero Point

To facilitate programming, programming personnel use work piece coordinate system (also called part coordinate system). A point is selected as the program zero point (This point is specified by G50 code and it is called program zero point), and a workpiece coordinate system is setup.

### 9.4 The Operation Procedures of Program Zero Return

1. Press  key to enter the Program Zero mode, the lower right corner of the screen displays “Machine Zero”.
2. Select X and Z axes that to be returned to program zero point.
3. The machine moves toward the program zero, and the axis stops with program zero indicator  lighting up after the axis returns to the program zero.

**Note:** G50 should be executed after system is powered on, otherwise, #021 alarm occurs.

## Chapter Ten System Communication

Serial terminal port software applies to **GSK980TB3/TB3-V** CNC system and bi-directional data conversion between computers.

- Operating environment: Windows 98/ Windows ME / Windows 2000/ Windows XP
- Interface standard: RS232
- Max. transmission rate: 115200 bit/s
- Optimum transmission rate: 9600 bit/s

Version characteristics:

A computer can communicate with a NC system only.

Maximum transmission rate 115200bit/s (Corresponding setting for the system is needed)

Hardware connection:

Before the file transmission, connect PC machine serial port to the NC system communication port.

Make sure connection is correct, and file transmission can be performed after the CNC operates normally.

### 10.1 Serial Port Communication

Serial port communication software is window interface, which is used for file transferring/receiving from PC end to CNC end. This software can be applied to Win98. WinMe. WinXP and Win2K.

#### 10.1.1 Program Start

Communication software program is performed directly. Example: Comm980TB3.exe. The interface is displayed after the program starts, refer to the following figure:

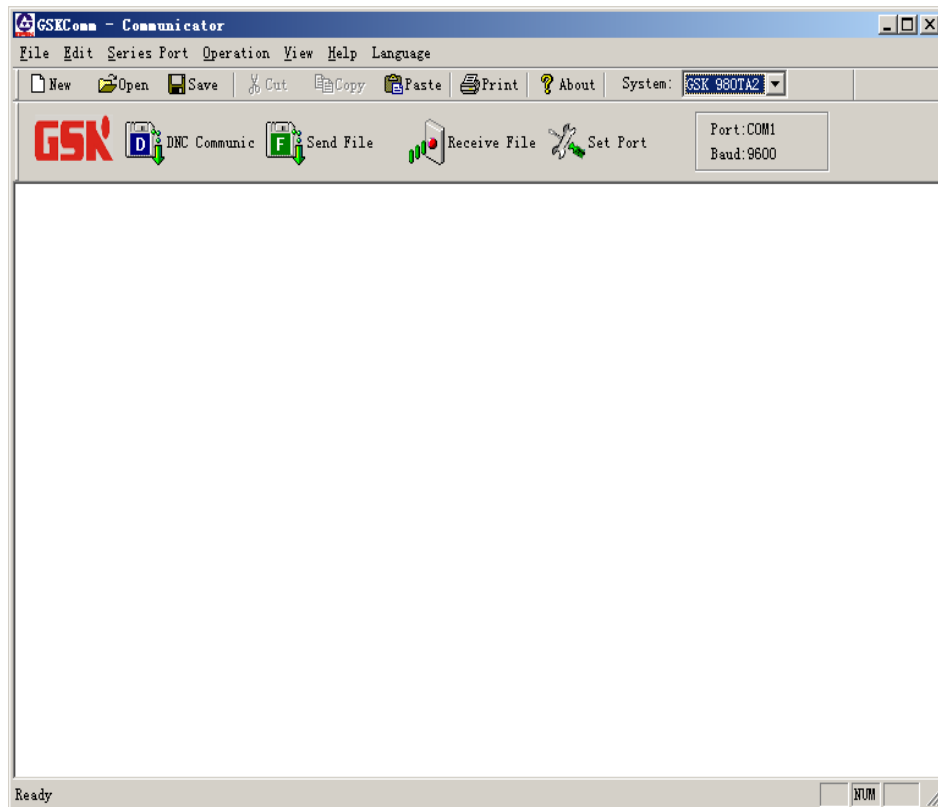


Fig. 10-1-1-1

### 10.1.2 Function Instruction

#### 1. File menu

File menu includes Add Files, Open, Save List, Print and Print Setting, Recently opened file lists, etc. functions.

#### 2. Edit menu

Edit menu includes Cut, Copy, Paste, Undo, Search, Replace, etc. functions.

#### 3. Serial port menu

This is mainly used for opening and setting serial port.

#### 4. Transmission mode menu

It includes DNC transmission mode (it is not used temporarily), File Sending Mode, File Receiving Mode.

#### 5. View menu

Display and concealment of both toolbar and status bar.

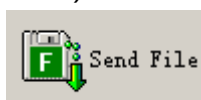
#### 6. Help menu

Version information for the software.

### 10.1.3 Software Usage

#### 1. Methods of sending files

- 1) Open and set a serial port. Baud rate can be set to 9600, 57600 or 115200.
- 2) After “send file” menu of the transmission mode menu is opened or the button send file



of the toolbar is pressed, the dialogue box will be popped up as below:

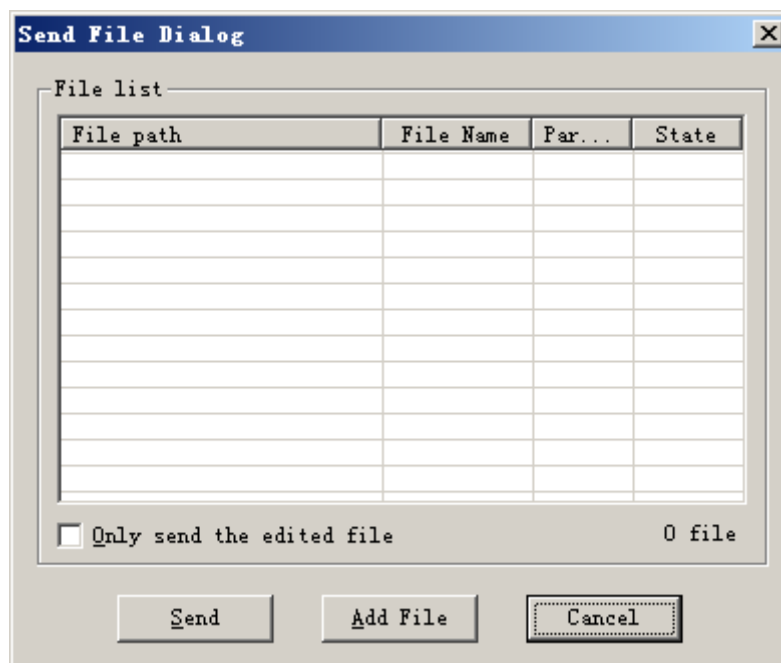


Fig. 10-1-3-1

- 3) Select “add files” button, the dialogue box “select zone” is displayed:



Fig. 10-1-3-2

Program files can be only sent to “user zone”, while the system configuration files to the system zone. Otherwise, the files can not be identified by the system. System manufacturer of machine manufacturer authorities are necessary for sending system configuration files or backup files. The corresponding password can be input on “password set” of the Setting page on the CNC end.

- 4) The dialogue box appears after zone is selected, press Shift or Control key to select multiple files: maximum 399 files can be selected.
- 5) Click “open” button after the files are selected, then return to “send file” dialogue box.
- 6) Names of the programs to be sent to the user zone must be headed with the letter “O” and followed by four-digit number. Otherwise, the dialogue box will prompt the user to change the name.

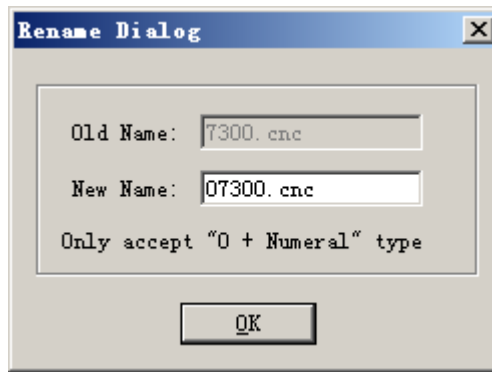


Fig. 10-1-3-3

7) Return to “send file” dialogue box, and click “send” key. The file is transferred and the dialogue box will be shown as bellow:

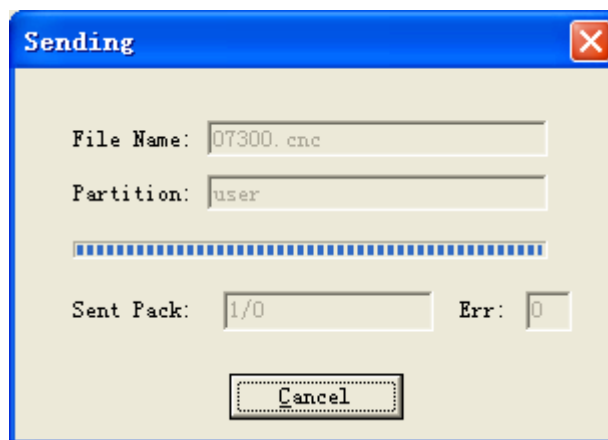


Fig. 10-1-3-4

8) Transmission complete

**Note:** If the program transmission is performed when the system works, it will affect normal operation of the system.

## 2. Methods of receiving files

1) Enter into the data interface of the setting page, Move the contents to the table of “data output” by red mark of the direction keys.

2) Open and set the serial port, baud rate can be set to 9600, 57600 or 115200.

3) Open “receive file” of the transmission menu or press receive file key , the dialogue box will be popped up as below:



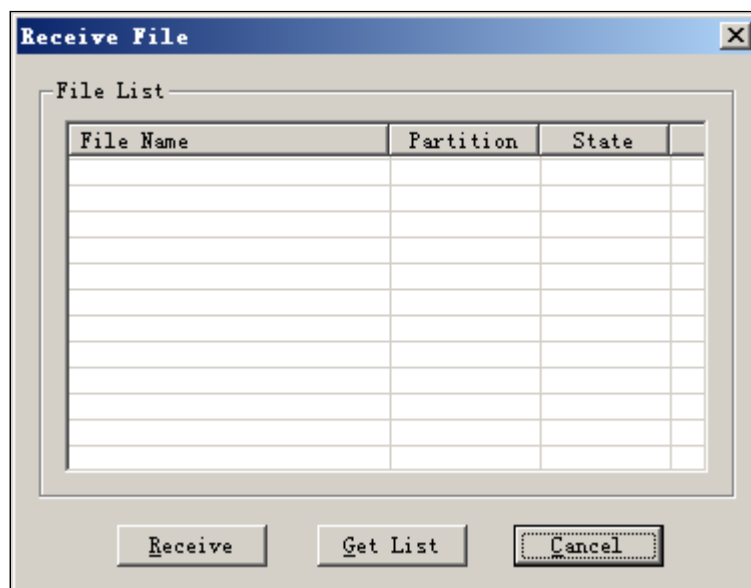


Fig. 10-1-3-5

- 4) Press “obtain list” key to get file list of the CNC.
- 5) Select the file to be transferred. Multiple files can be selected by pressing Shift or Ctrl key.
- 6) Click “start receiving” key, select file storage address and then the dialogue box is popped up as bellow:

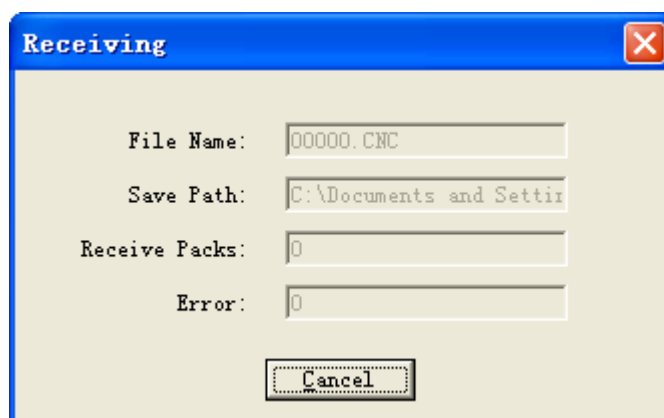


Fig. 10-1-3-6

- 7) Transmission completes.

Note: If the program transmission is performed when the system works, it will affect normal operation of the system.

#### 10.1.4 Communication between CNC and CNC

Two CNC systems can communicate with each other. Take 980TB3 for an example as bellow.

1. Mutual program communication between two CNC systems.
  - (1) Connect two 980TB3 systems with communication line.
  - (2) After two 980TB3 systems are powered on, set identical baud rates for them, e.g. 9600.
  - (3) In the system 980TB3 to send programs, press edit key to enter the edit mode, select the program name (Supposing it is O8888), press the address key “O” and the numerical key, then wait for the system displaying communication success and the communication completes.

## 2. Mutual parameter communication between two CNC systems

Connect two GSKGSK980TB3 systems with communication line, and turn on parameter switch after the systems power on. The following operations are performed on the system to send parameters.

Select the Edit mode and enter the parameter interface. Press  key on the panel, and now “communicating” is displayed in the lower part of the screen (If the CNC systems are not connected well, alarm “the port is not connected” occurs in the lower part of the interface). Communication is finished after the characters disappear. If alarm “communication error” occurs, please check whether the baud rates of the two GSK980TB3 systems are consistent or lower the transmission baud rates of the ports. After communication, the parameters of the GSK980TB3 system that receives parameters are updated, now it is necessary to power on again. When the screw-pitch compensation parameters to be sent, enter the screw-pitch compensation parameter interface, and the operation steps are the same as that of the parameter transmission.

## 10.2 USB Communication

### 10.2.1 Overview and Cautions

1. Because the U-disk files is entirely separated from the CNC system, you had better not enter into the U-disk system in operation. Otherwise, the problem of work piece damage may occur.
2. The U-disk file system supports repeatedly hot plug and play. However, do not plug the U-disk before power-on to avoid failing to identify the U-disk. Plug it after entering the U-disk operation interface.
3. To avoid data in U-disk is not performed completely, pull out the U-disk when the indicator is not flash or wait a little while.
4. Only txt file with a name “O+ four digits” can be displayed, e.g. “O0001”.

### 10.2.2 Enter U-disk

1. After the CNC system is powered on, press down “setting” key, and press “D” to turn on the parameter switch, then the system enters the state of alarm. By pressing edit key, the system is shifted to the edit mode, and press the program key twice, the system enters to USB operation interface.
2. Plug in U-disk after the system enters the U-disk program.

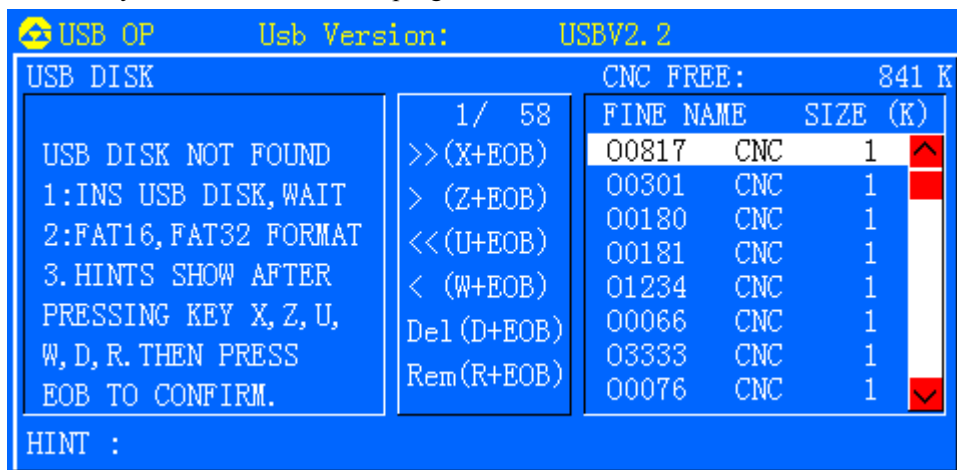


Fig. 10-1-2-1

### 10.2.3 Procedures

Plug in U-disk, the system will automatically read U-disk, all O????CNC programs on root directory of the USB disk and files: "PARA3.txt", "PARA4.txt ", "WORM2.txt ", "WORM3.txt ", "WORM4.txt ". By pressing left, right arrows, the cursor can be shifted between CNC memory and USB memory. And the file on the current memory can be selected by pressing up and down keys.

#### 1. Copy CNC program from U-disk to the user disk

##### 1.1 Copy all CNC programs from U-disk to the user disk

- a. Press left and right arrows to display U-disk files.
- b. After pressing “X” key, the interface prompts that: copy all files from USB to the CNC system. This operation can be cancelled by pressing left and right arrows. Press the <enter> key to start the duplication. If same file names exist, the system will prompt that: files have the same name, O???? CNC interchange (IN) ,cancel (CAN) , all (ALL) . Select the one needed.
- c. After duplication is finished, the system will prompt: copy completed.

##### 1.2 Copy a CNC program from U-disk to the user disk

- a. Press left and right arrows to display U-disk files.
- b. Select files on the U-disk by pressing up and down keys.
- c. After pressing “Z” key, the interface prompts that: copy the files selected from USB to the CNC system. This operation can be cancelled by pressing left and right arrows. Press the <enter> key to start the duplication. If same file names exist, the system will prompt that: “files have the same name” O???? CNC interchange (IN) ,cancel (CAN) , all (ALL) . Select the one needed.
- d. After duplication is finished, the system will prompt: copy completed.

#### 2. Delete files from the U-disk

##### 2.1 Delete all files from U-disk

- a. Press left and right arrows to display U-disk files.
- b. After pressing “R” key, the interface prompts that: “delete all files from USB”. This operation can be cancelled by pressing left and right arrows. Press the <enter> key to delete the files.
- c. There is no files in the U-disk after they are deleted.

##### 2.2 Delete a file from the U-disk

- a. Press left and right arrows to display U-disk files.
- b. Select CNC files in the U-disk by pressing up and down keys.
- c. After pressing “R” key, the interface prompts that: “delete the files selected in the USB”. This operation can be cancelled by pressing left and right arrows. Press the <enter> key to delete the files.
- d. This file is not in the U-disk after it is deleted.

#### 3. Copy CNC programs from the system user disk to the U-disk

##### 3.1 Copy all CNC programs from the user disk to the U-disk

- a. Press left and right keys to the lists of the CNC.
- b. After pressing “U” key, the interface prompts that: copy all CNC programs from the user disk to the U-disk. This operation can be cancelled by pressing left and right arrows. Press the <enter> key to copy the files. If same file names exist, the system will prompt that: files have the same name, O???? CNC interchange (IN) ,cancel (CAN) , all (ALL) . Select the one needed.

- c. After duplication is finished, the system will prompt: copy completed.

### 3.2 Copy a CNC program from user disk to the U-disk

- a. Press left and right keys to the lists of the CNC.
- b. Press up and down keys to select CNC files in the CNC memory.
- c. After pressing “W” key, the interface prompts that: copy the selected program from the CNC to the USB system. This operation can be cancelled by pressing left and right arrows. Press the <enter> key to copy the files. If same file names exist, the system will prompt that: files have the same name, O???? CNC interchange (IN) ,cancel (CAN) , all (ALL) . Select the one needed.
- d. After duplication is finished, the system will prompt: copy completed.

## 4. Delete files from the system user disk

### 4.1 Delete files from the system user disk

- a. Press left and right keys to the lists of the CNC.
- b. After pressing “R” key, the interface prompts that: delete all files from the CNC. This operation can be cancelled by pressing left and right arrows. Press the <enter> key to delete the files.
- c. There is no files in the U-disk after they are deleted.

### 4.2 Delete a file from the user disk

- a. Press left and right keys to the lists of the CNC.
- b. Select CNC files in the CNC memory by pressing up and down keys.
- c. After pressing “D” key, the interface prompts that: delete the selected file from the CNC. This operation can be cancelled by pressing left and right arrows. Press the <enter> key to delete the file.
- d. This file is not in the CNC memory after it is deleted.

### 10.2.4 U-disk Removal

- 1. Pull out the U-disk until the indicator is not flash.
- 2. Press the screen operation keys to enter the system, or press reset key to cancel the alarm and enter the CNC system.

## PART FOUR CONNECTION



## Chapter One System Configuration and Installation

### 1.1 System Configuration

GSK980TB3/TB3-V systems include many units as follows, see figure 1-1-1 for the system configuration.

- (1) GSK980TB3/TB3-VCNC system
- (2) Additional operation panel
- (3) Digital AV servo drive unit (Stepping motor drive unit)
- (4) Servo motor (Stepping motor)
- (5) Isolation transformer

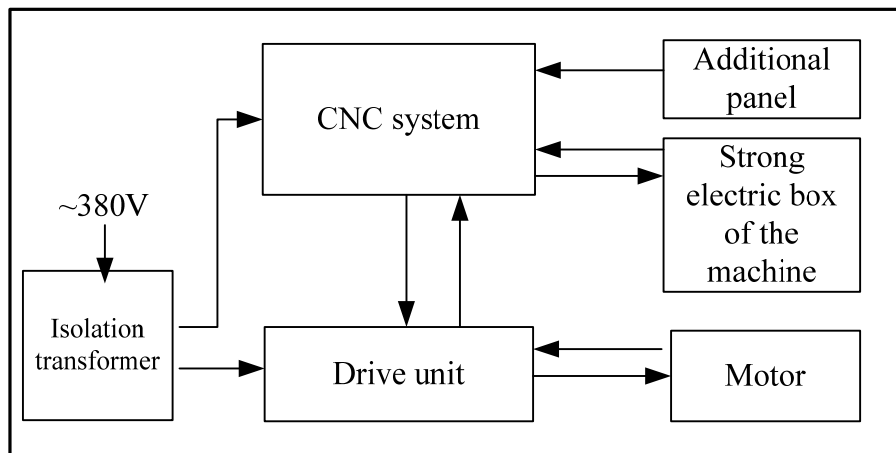


Fig. 1-1-1

### 1.2 System Installation

Check whether the units such as CNC system, drive unit, motor and photoelectric encoder etc. to be installed are complete, intact and the one you ordered.

The system must be installed securely and keep a certain space around it for heat dissipation. The CNC should be mounted at a place where is convenient for operation without chip and cooling.

Strong and weak current shall be separated. The isolation transformer provides power for the CNC and the drive unit. It should be separated from the strong current part. Each signal line shall keep as far away from the AC contactor to reduce interference. It is better to connect the photoelectric encoder, limit signal and emergency stop signal to the system directly, but not to strong electric box. Power supply must be strictly grounded.

Fix each connector and screw securely. It is not allowed to plug in or pull out the connector after system power on.

Prevent the panel from damaging by hard and sharp tools when installing the CNC system. When painting, remove the operation panel to avoid the paint stain it.

Ensure there is no strong current, strong magnetic interference source around the CNC system, and keep it far away from inflammable and explosive materials or other dangerous objects.



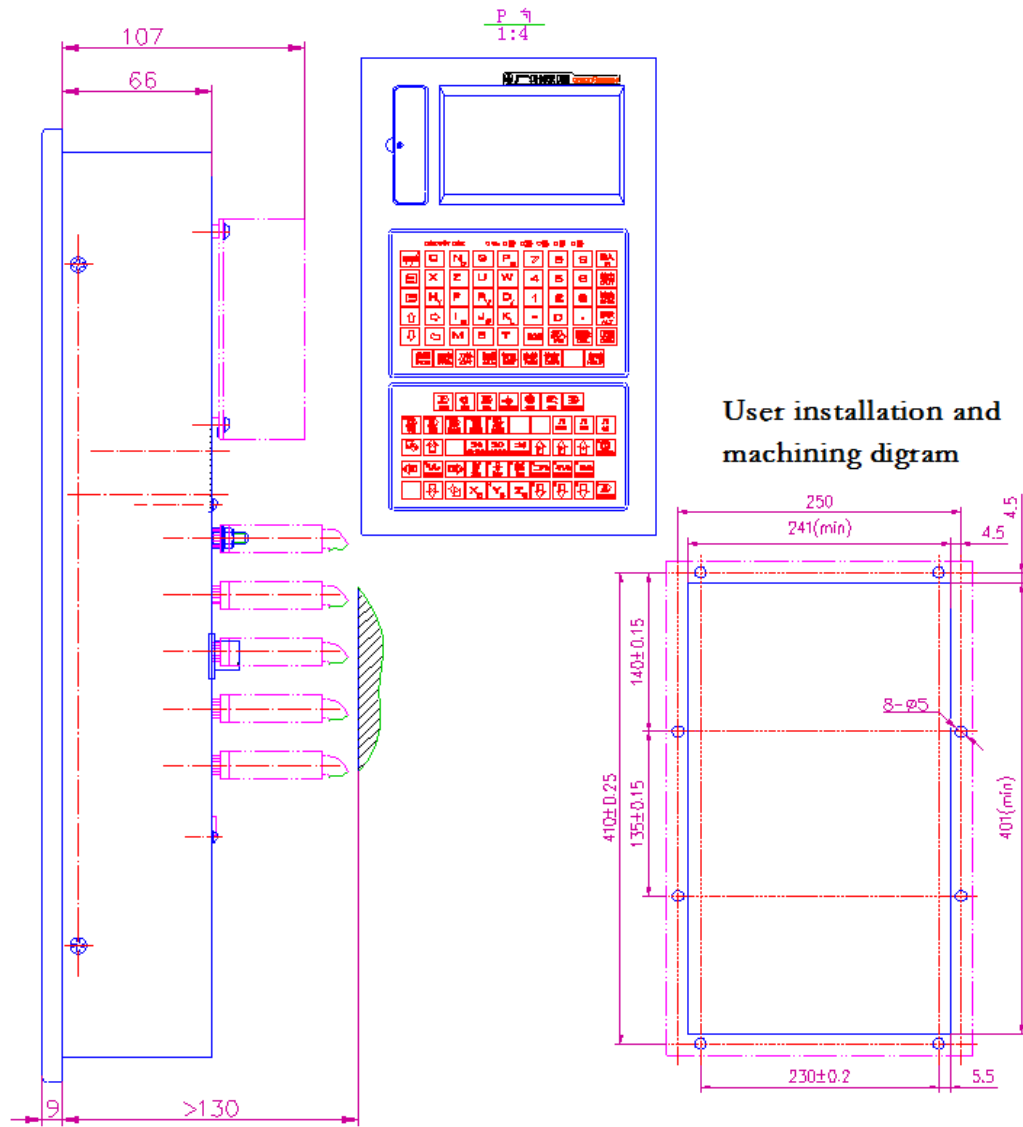


Fig. 1-3-2 Mounting dimension of the system GSK980TB3-V

#### 1.4 Additional Panel

User can select additional panel for this system and define the functions such as emergency stop, program lock, system power on/off, feed hold, cycle start, MPG etc. for the expansion hole of the panel. The optional accessories for the system GSK980TB3/ TB3-V are as follows:

Communication elements: communication cable 5m×1, communication software installation CD ×1.

Electronic MPG: MPG60-T1-100B/05 (match with AP01 additional panel) or MPG80-T1-100B/05 (match with AP02 additional panel)

Additional panel: AP01 (aluminum alloy 420mm×71mm) , it can be fixed bellow the panel

AP02 (aluminum alloy 100mm×260mm) , it can be fixed at the side of the panel

Emergency stop button: LAY3-02ZS/1 (GSK980TB3/TB3-V have fixed the emergency button)

Non-self-lock button: KH-516-B11 (Green or red)

Self-lock button: KH-516-B21 (Green or red)

**Note 1:** Communication function is the standard function configuration, but communication elements are optional

accessories.

**Note 2: Optional accessories are supplied as product accessories (unset and unconnected). Please note on the order if installation and connection are necessary.**

### Additional panel AP01

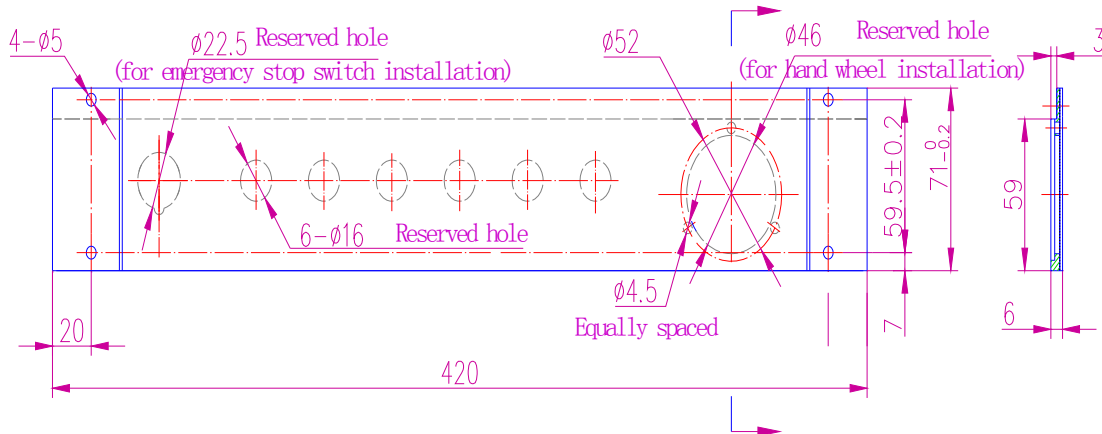


Fig. 1-4-1

### Additional panel AP02

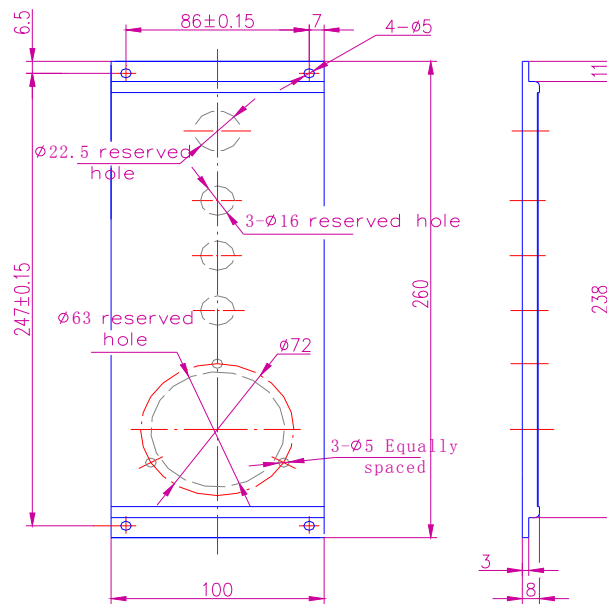


Fig. 1-4-2

## Chapter Two Equipment Connection

### 2.1 System Interfaces

#### 2.1.1 Interface Layout

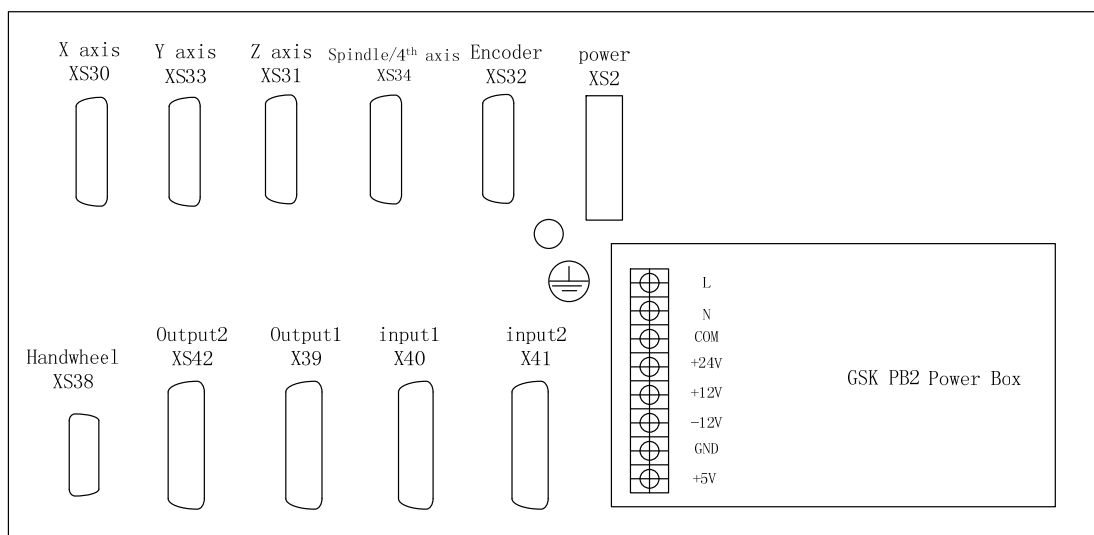


Fig.2-1-1-1 GSK980TB3 back view

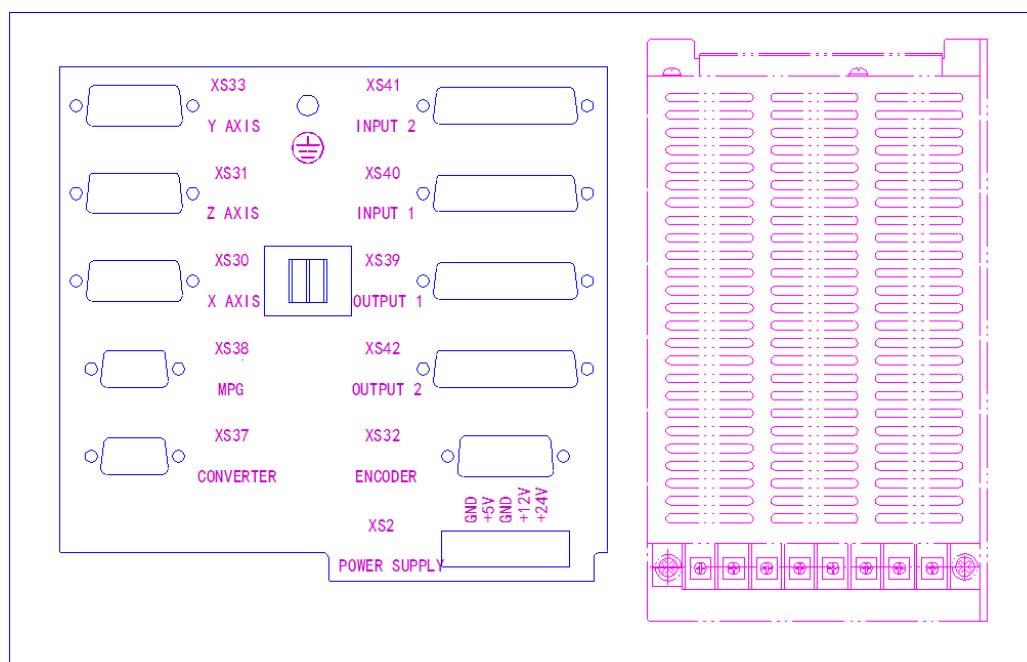


Fig.2-1-1-2 GSK980TB3-V back view

### 2.1.2 Explanations for Interfaces

- 1) Power interface (+5V, +24V, 0V)  
5 terminals for the system and power interface.
- 2) XS30, XS31, XS33 are drive interface of the X, Z, Y axis respectively.  
15-core D type socket (female)
- 3) XS32 spindle encoder interface (incremental pulse encoder, 1024 or 1200 wires)  
15-core D type socket (male)
- 4) XS36 general-purpose PC machine interface (RS232 port)  
9-core D type socket (female)
- 5) XS34 spindle interface  
26-core D type socket (female)
- 6) XS38 MPG interface  
26-core D type socket (male)
- 7) XS39, XS40 input/output interface  
XS39 is 25-core D type socket (female), XS40 is 25-core D type socket (male)
- 8) XS41, XS42 extended input/output interface  
XS41 is 25-core D type socket (male), XS40 is 25-core D type socket (female)

### 2.1.3 Connection Diagram

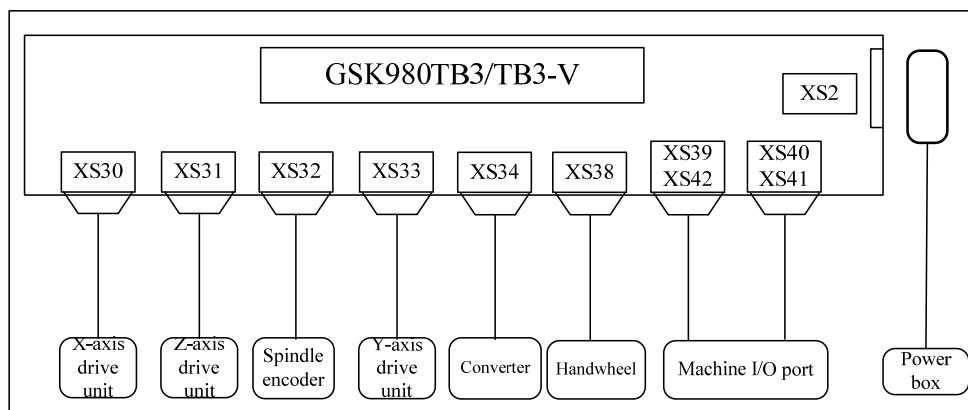


Fig. 2-1-3-1

## 2.2 Connection of the System to the Drive Unit

### 2.2.1 Connection to the Drive Unit

- 1) Interface diagram of PC side

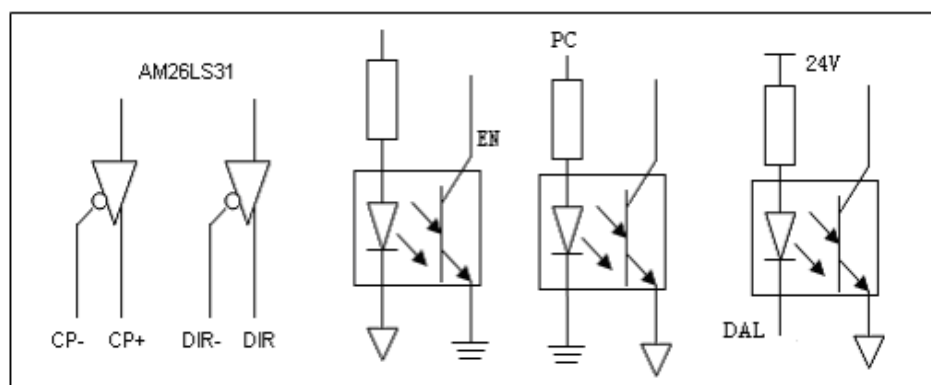


Fig. 2-2-1-1

## 2) Definition of the interface signal

| XS30: DB15F (X-axis) |       |    |       | XS31: DB15F (Z-axis) |       |    |       |
|----------------------|-------|----|-------|----------------------|-------|----|-------|
| 1                    | XCP+  | 9  | XCP-  | 1                    | ZCP+  | 9  | ZCP-  |
| 2                    | XDIR+ | 10 | XDIR- | 2                    | ZDIR+ | 10 | ZDIR- |
| 3                    | XPC   | 11 | 0V    | 3                    | ZPC   | 11 | 0V    |
| 4                    | +24   | 12 | +5V   | 4                    | +24   | 12 | +5V   |
| 5                    | XDALM | 13 | +5V   | 5                    | ZDALM | 13 | +5V   |
| 6                    | *XSET | 14 | 0V    | 6                    | *ZSET | 14 | 0V    |
| 7                    | XEN   | 15 | 0V    | 7                    | ZEN   | 15 | 0V    |
| 8                    | 0V    |    |       | 8                    | 0V    |    |       |

| Signal       | Description             |
|--------------|-------------------------|
| nCP+, nCP-   | Code pulse signal       |
| nDIR+, nDIR- | Code direction signal   |
| nPC          | Zero signal             |
| nDALM        | Drive unit alarm signal |
| nEN          | The system ready signal |
| nSET         | Setting signal          |

Fig. 2-2-1-2

### 2.2.2 Signal Explanation

#### 1. Mono-pulse movement commands signal

XCP+, XCP-, ZCP+, ZCP- are instruction pulse signal, XDIR+, XDIR-, ZDIR+, ZDIR- are movement direction signal. Two groups of signals are differential output signals.

Connection diagram is as follows:

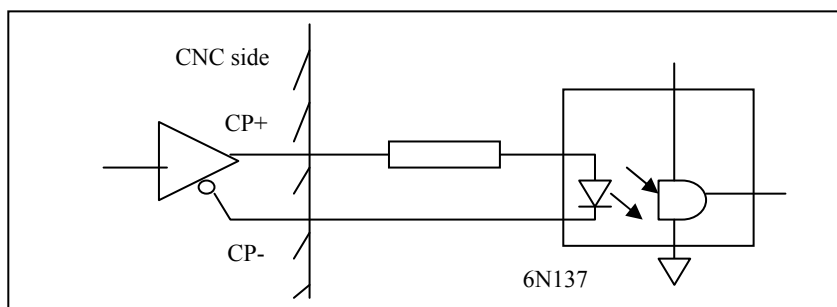


Fig. 2-2-2-1

#### 2. Alarm signal of the drive unit (input)

The receive mode of the signal at the system side is as follows. By setting the parameter (008), it is able to set whether low level "0" or high level "1" is drive unit fault.

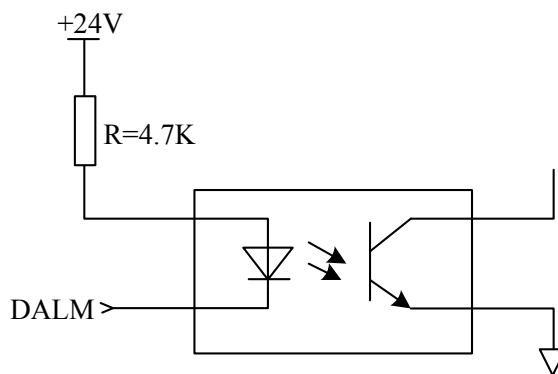


Fig. 2-2-2-2

### 3. CNC system ready signal EN (contact output)

Closed contact indicates the CNC system is ready. When the system alarm, the contact signal will be cut off.

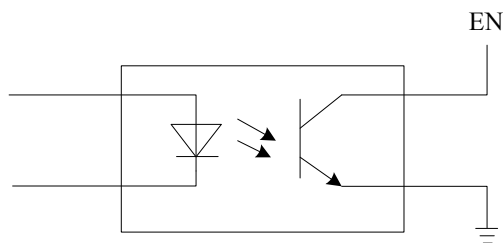


Fig. 2-2-2-3

### 4. Set signal \*SET (output)

When there are movement commands output from the system, it is high level signal. When there is no movement command output from the system, it is low level signal.

### 5. Signal PC is used for returning to the reference point

The receive mode of the signal at the system side is as follows.

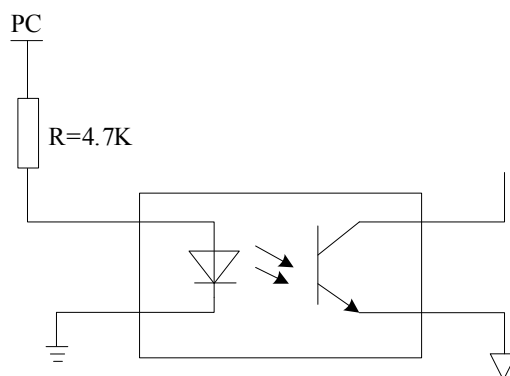


Fig. 2-2-2-4

**Note 1:** If PC signal output by the drive unit is 5V, wire jumper makes SR3028, SR3030, SR3032 (510Ω) on. If PC signal output by the drive unit is 24VV, wire jumper makes SR3027, SR3029, SR3031 (4.7KΩ) on.

**Note 2:** The user should provide the PC signal, which waveform is as follows.

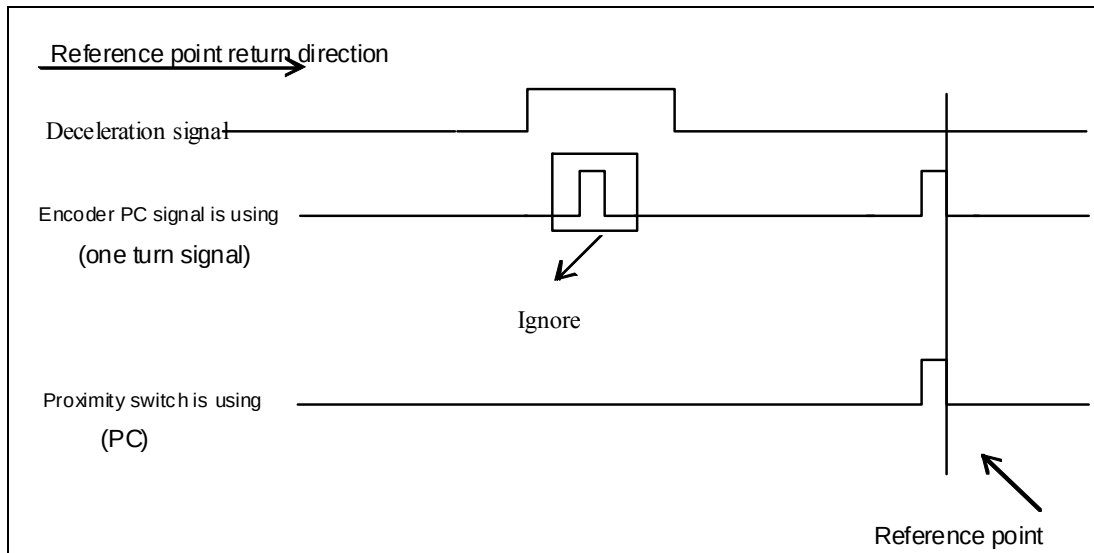


Fig. 2-2-2-5

- 6) Connection methods for a proximity switch that is taken as a deceleration switch and a zero return switch.

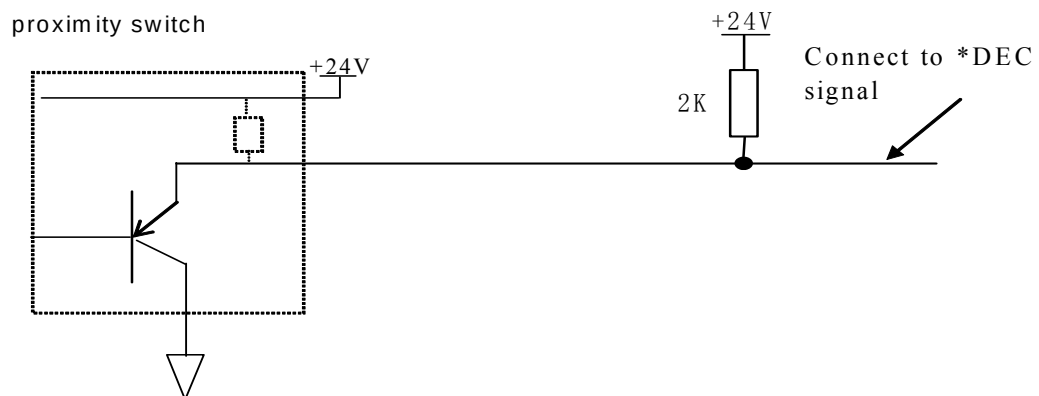
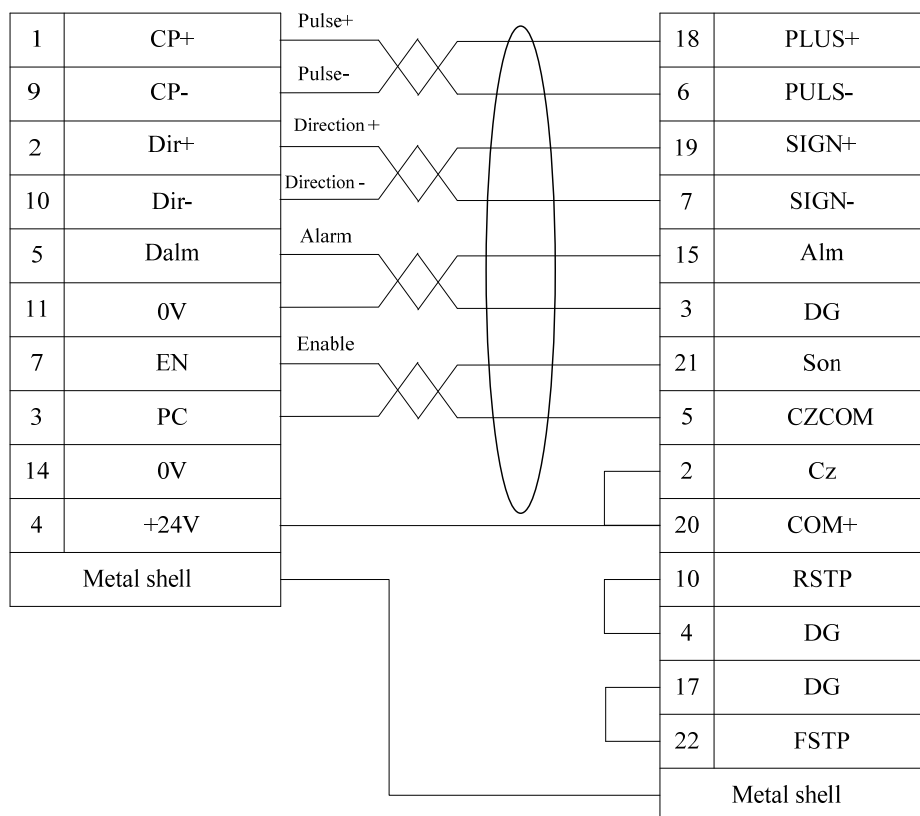


Fig. 2-2-2-6

**Note:** When this zero return mode is used, ZCX, ZCZ of the parameter should set to “1”.

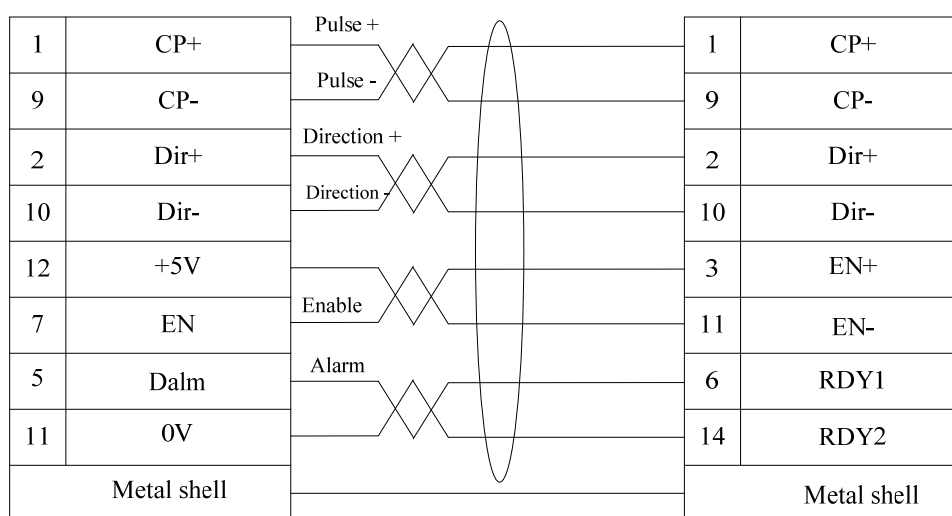
## 2.2.3 Connection Between the System and the Drive Unit

GSK 980TB3/TB3-V connect to DA98 drive unit  
 GSK 980TB3/TB3-V ( X, Y,Z axis )  
 X30,X31,X33



GSK 980TB3\TB3-V connect to DY3 drive unit

GSK 980TB3\TB3-V ( X, Y,Z axis )  
 X30,X31,X33



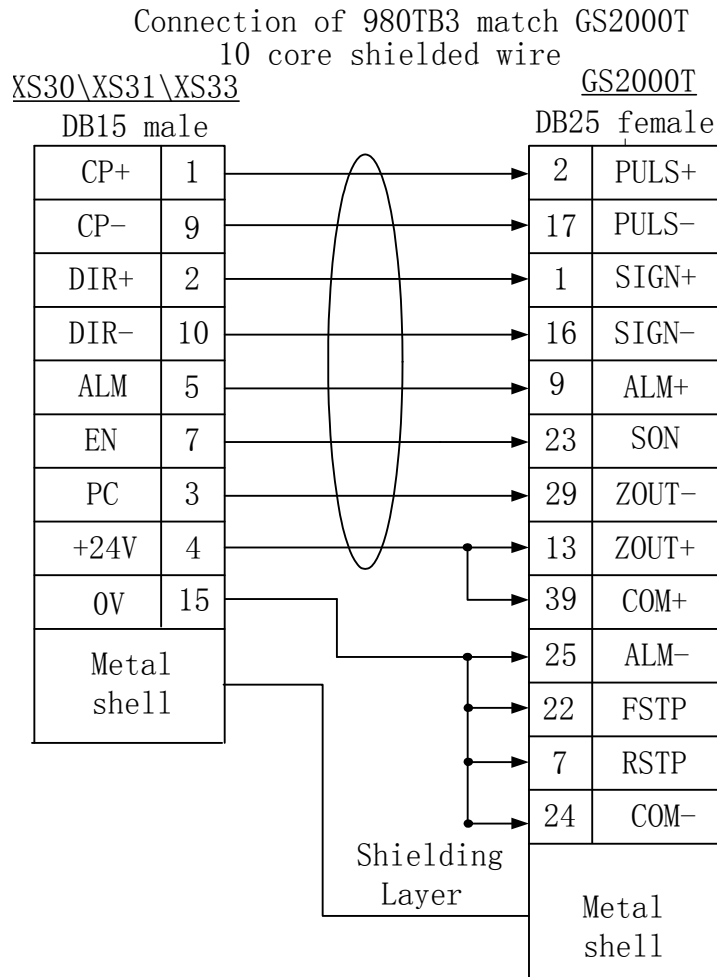


Fig. 2-2-3-1

### 2.3 Connection to the Spindle Encoder

1024-line or 1200-line incremental position encoder can be used with the system. \*PCS, PCS, \*PBS, PBS, \*PAS, PAS bellow correspond to Z, Z, B, B, A, A respectively.

XS32: DB15F (Spindle encoder)

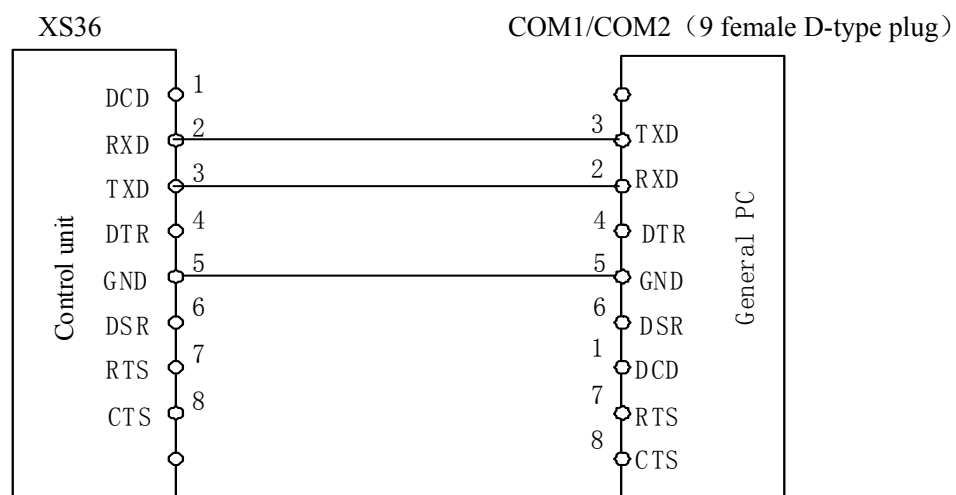
|   |      |    |     |
|---|------|----|-----|
| 1 |      | 9  |     |
| 2 |      | 10 |     |
| 3 | *PCS | 11 | 0V  |
| 4 | PCS  | 12 | +5V |
| 5 | *PBS | 13 | +5V |
| 6 | PBS  | 14 | 0V  |
| 7 | *PAS | 15 | 0V  |
| 8 | PAS  |    |     |

| Signal    | Explanation           |
|-----------|-----------------------|
| PAS, *PAS | Encoder A-phase pulse |
| PBS, *PBS | Encoder B-phase pulse |
| PCS, *PCS | Encoder Z-phase pulse |

Fig. 2-3-1

## 2.4 RS232-C Standard Serial Port (Optional)

The CNC system can communicate with PC machine via RS232-C serial port (Special communication software shall be installed). Connection diagram is as follows.



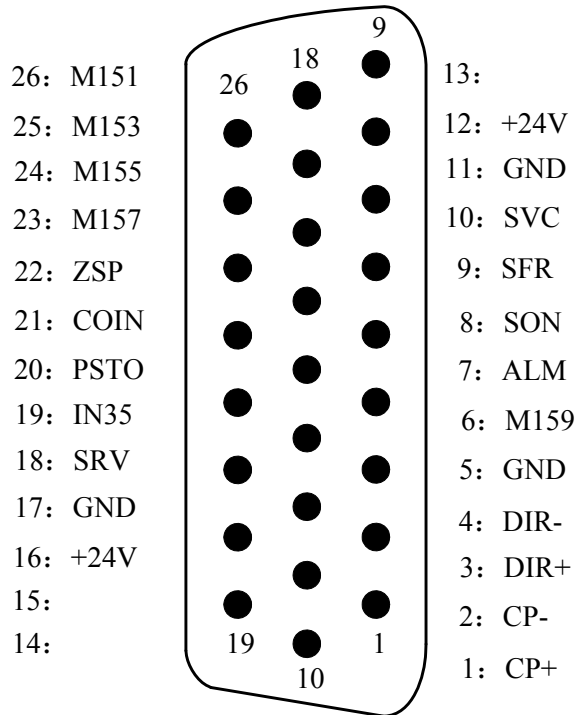
| Signal | Explanation    |
|--------|----------------|
| RXD    | Data receiving |
| TXD    | Data sending   |

Fig. 2-4-1

**Note:** Weld the shielding layer to the metal of the plug.

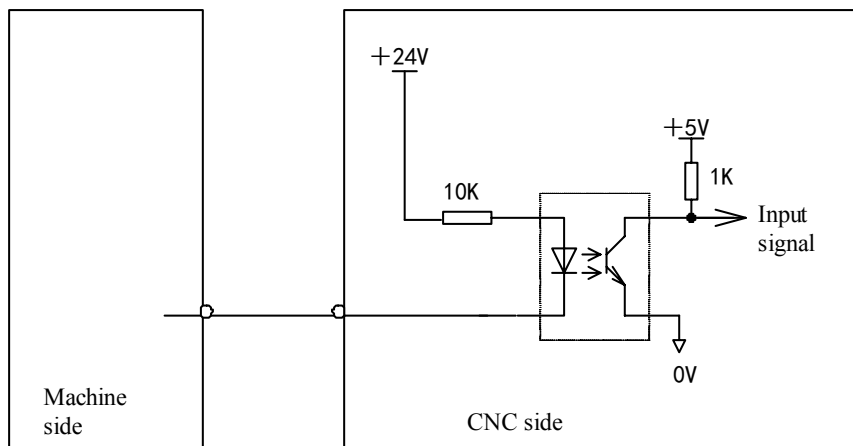
## 2.5 Spindle Interface

XS34 spindle/4<sup>th</sup> interface  
DB26 female

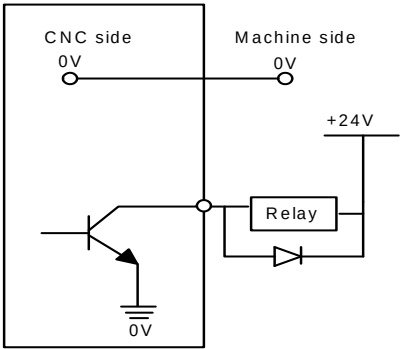


| Signal      | Explanation                                 |
|-------------|---------------------------------------------|
| CP4+、CP4-   | Spindle pulse signal                        |
| DIR4+、DIR4- | Spindle direction signal                    |
| SVC         | Analog voltage output                       |
| M151        | Spindle orientation                         |
| M153        | Zero speed clamp                            |
| M155        | Speed/pos switch                            |
| M157        | M157                                        |
| M159        | M159                                        |
| SON         | Spindle enable signal                       |
| SFR、SRV     | Spindle CW/CCW                              |
| ZSP         | Spindle zero speed check                    |
| COIN        | Spindle orientation finish                  |
| PSTO        | Spindle speed/pos switch finish             |
| IN35        | Spindle clamp input                         |
| ALM         | 4 <sup>th</sup> axis/spindle abnormal alarm |
| +24V        | +24V                                        |
| GND         | 0V                                          |

Input signal circuit of spindle interface (XS34) :



Onput signal cicuit of spindle interface (XS34) :



2.5.1 Connection drawing of 980TB3 and normal relay

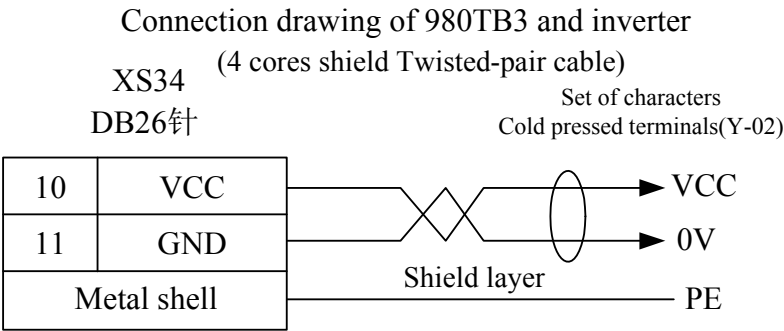


Fig.2-5-1

Note: Weld the shielding layer to the metal of the plug.

## 2.5.2 Connection drawing of 980TB3 and spindle driver DAP03(require position and speed control function)

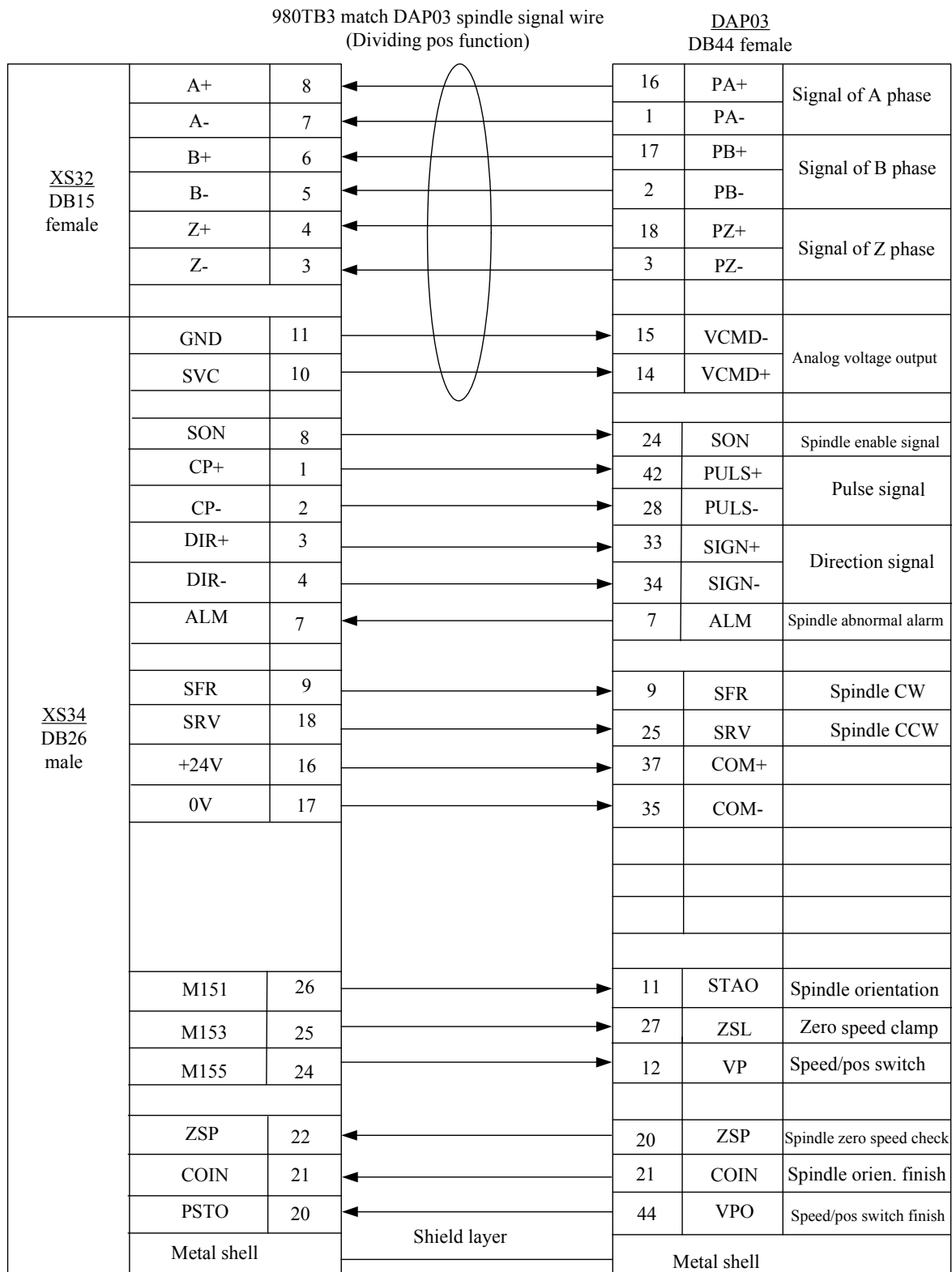


Fig.2-5-2

### 2.5.3 Connection drawing of 980TB3 and spindle driver GS3000Y-N (require position and speed control function)。

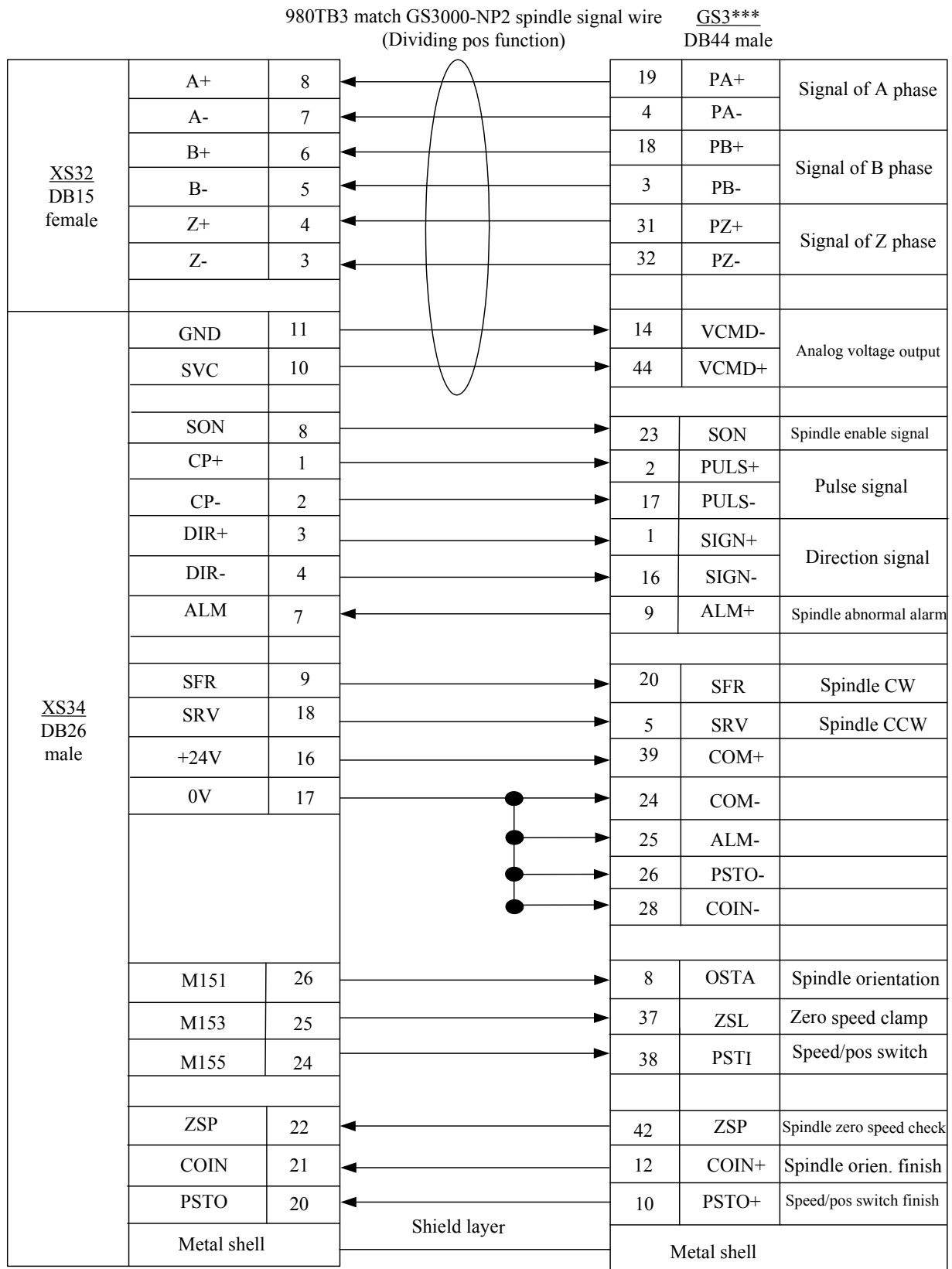
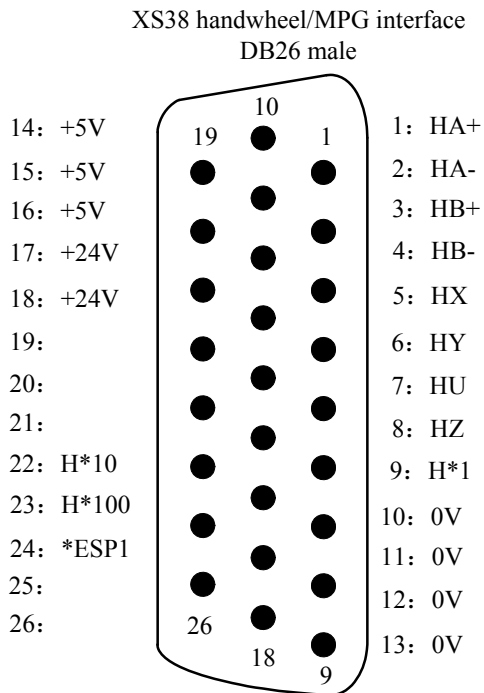


Fig.2-5-3

## 2.6 MPG Interface



| Signal   | Explanation           |
|----------|-----------------------|
| HA+, HA- | Signal of MPG A phase |
| HB+, HB- | Signal of MPG B phase |
| HX       | X handwheel axial     |
| HY       | Y handwheel axial     |
| HU       | U handwheel axial     |
| HZ       | X handwheel axial     |
| H*1      | Increment*1           |
| H*10     | Increment*10          |
| H*100    | Increment100          |
| *ESP1    | Emergency stop signal |
| +24V     | +24V                  |
| +5V      | +5V                   |
| 0V       | 0V                    |

### 2.6.1 Normal MPG connetciton drawing

Connection drawing of GSK 980TB3 and single MPG

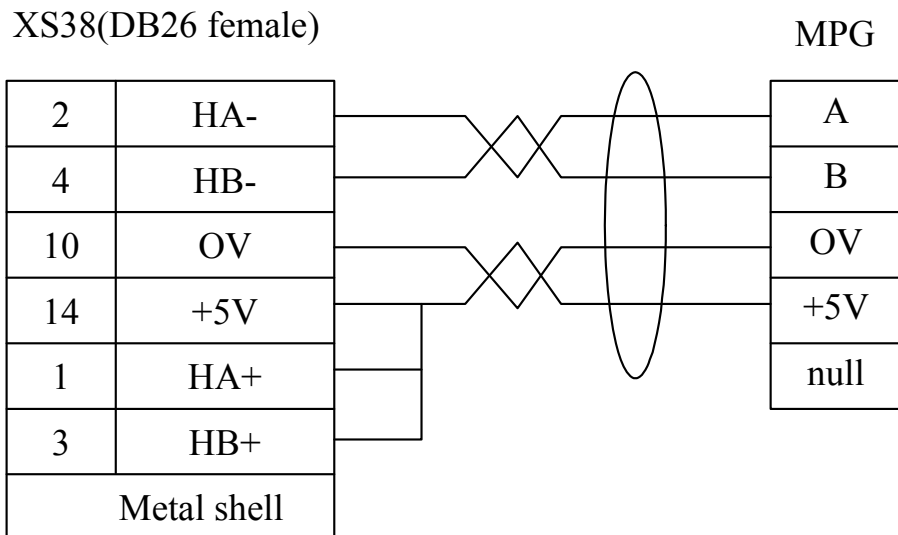
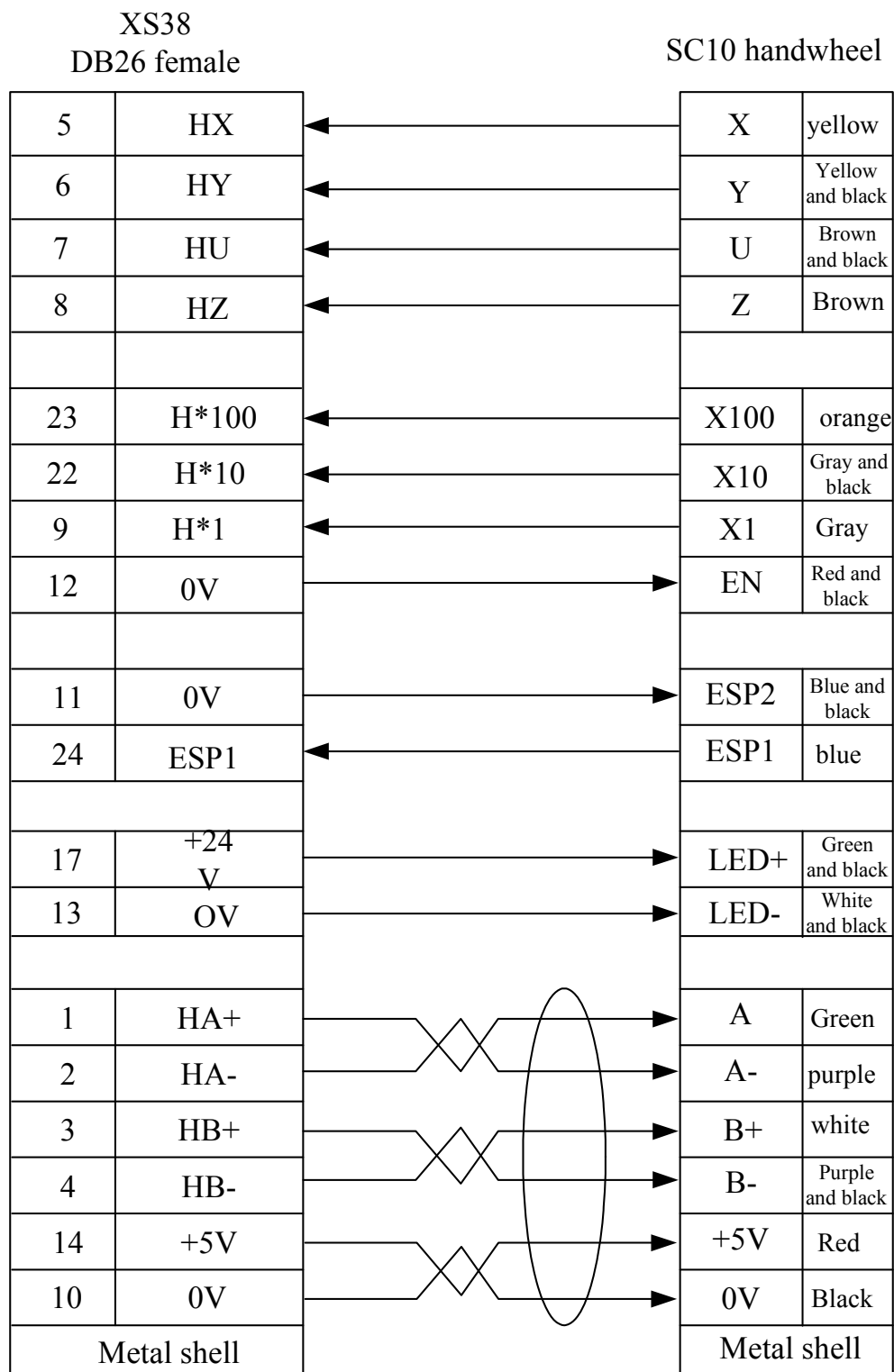


Fig.2-6-1

## 2.6.2 Connection drawing of handwheel (match MPG GSKSC10)

Connection drawing of GSK980TB3 and handwheel GSKSC10



Handwheel parameter set:

1. set No.1 sys parameter BIT3 to be 1, select MPG mode.
2. set No.7 sys parameter BIT6 to be 1, select external MPG mode.
3. when match row tool post, it is need to set No.81 to be 1.

## 2.7 Connection of the Power Interface

GSKPB2 power is applied in the systems GSKGSK980TB3/TB3-V, which involves three groups of input voltage:

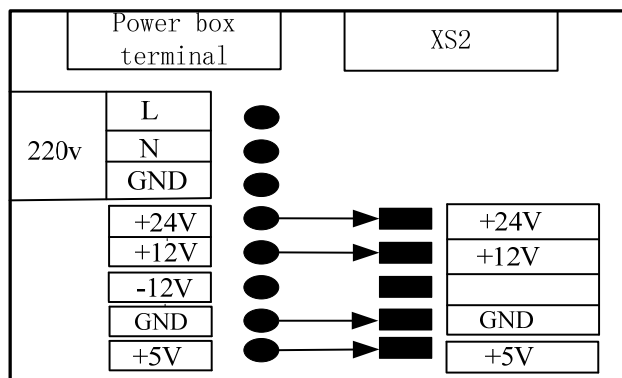


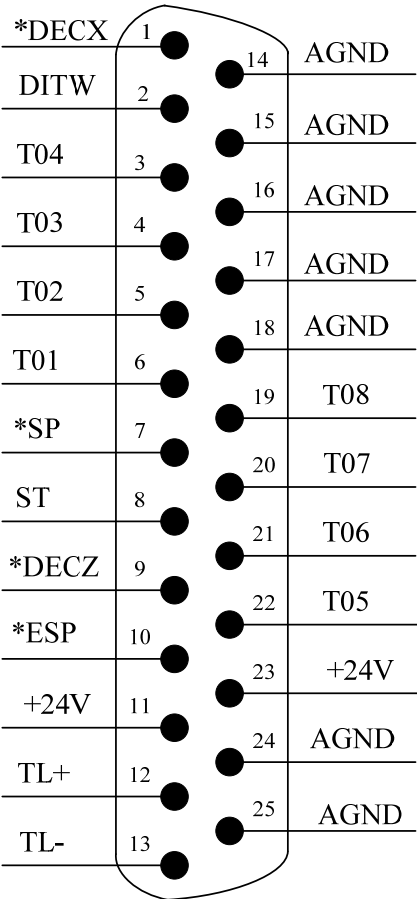
Fig. 2-7-1

Chapter Three I/O Interface of Machine Tool

3.1 List of the Machine Tool I/O Interface

3.1.1 Standard I/O Interface of GSK980TB3/TB3-V

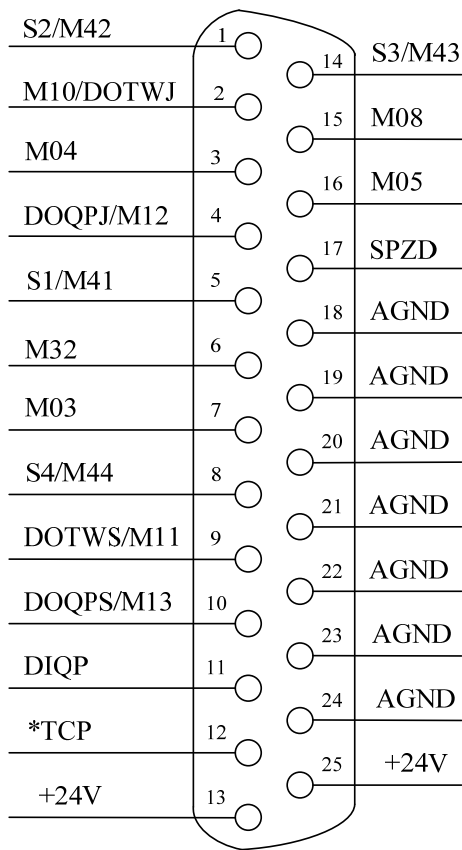
XS40 Input signal interface  
(male)



| Signal      | Explanation                            |
|-------------|----------------------------------------|
| *DECX、*DECZ | X, Z axis reference return DEC. signal |
| DITW        | Tail stock control signal              |
| T01~T08     | Tool number signal                     |
| *SP         | Feed hold signal                       |
| ST          | Cycle start signal                     |
| *ESP        | Emergency stop signal                  |
| TL+、TL-     | Tool post CCW, CW signal               |
| +24V        | +24V                                   |
| AGND        | 0V/ common terminal                    |

Fig. 3-1-1-1

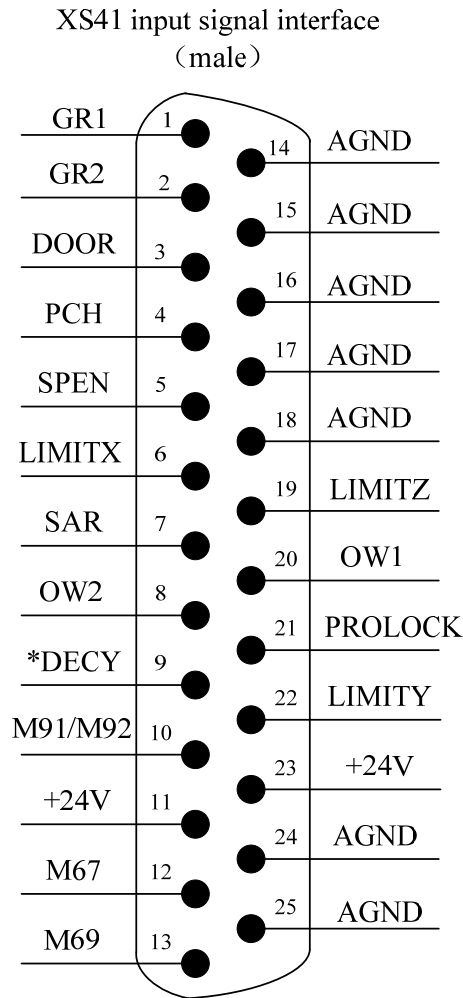
XS39 output signal interface  
(female)



| Signal      | Explanation                        |
|-------------|------------------------------------|
| S1~S4       | Spindle gear signal                |
| M41~M44     | Spindle auto gearing signal        |
| M03、M04、M05 | Spindle CCW, CW, stop signal       |
| M08         | Coolant ON                         |
| M32         | Lubricant ON                       |
| M10、M11     | Tail stock forward/ retreat        |
| M12、M13     | Clamp/ release signal              |
| SPZD        | Spindle break signal               |
| *TCP        | Tool post clamp in-position signal |
| DIQP        | Clamp control signal               |
| +24V        | +24V                               |
| AGND        | 0V/ common terminal                |

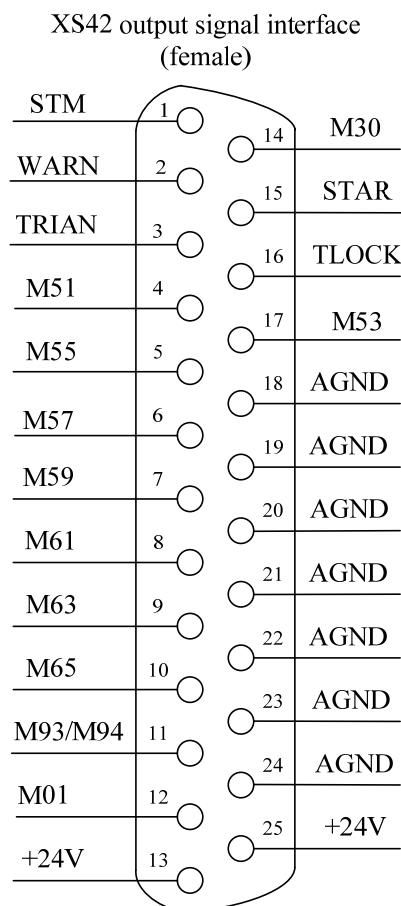
Fig. 3-1-1-2

3.1.2 Expansion I/O Interface of GSK980TB3/TB3-V



| Signal               | Explanatiion                        |
|----------------------|-------------------------------------|
| GR1、GR2              | Gearing in-position signal          |
| DOOR                 | Safety door detection signal        |
| PCH                  | Pressure detection signal           |
| SPEN                 | Feed enable signal                  |
| SAR                  | Spindle rotation enable signal      |
| LIMITX、LIMITZ、LIMITY | Hardware limit signal               |
| OW1、OW2              | External alarm signal               |
| PROLOCK              | Program lock signal                 |
| *DECY                | Y-axis reference return DEC. signal |
| M91/M92              | Expansion user input signal         |
| M67~M69              | Expansion M code signal             |
| +24V                 | +24V                                |
| AGND                 | 0V/ common terminal                 |

Fig. 3-1-2-1



| Signal     | Explanation                                                |
|------------|------------------------------------------------------------|
| STM        | Auto running signal                                        |
| M30        | Output alarm signal                                        |
| WARN       | Machining completion signal                                |
| STAR、TRIAN | Star triangle start signal                                 |
| TLOCK      | Tool post clamping signal (match with LIO SHING tool post) |
| M51~M65    | Expansion M code signal                                    |
| M93/M94    | Expansion user input signal                                |
| M01        | Expansion user input signal                                |
| +24V       | +24V                                                       |
| AGND       | 0V/ common terminal                                        |

Fig. 3-1-2-2

**Note 1:** Various functions can be defined to some of the input and output interfaces, and they are indicated by “/” sign in the above table.

**Note 2:** If the output signal is through on to COM, the output function is active. In the contrary situation, the output function is inactive.

**Note 3:** If the input signal is through on to +24V, the input function is active. In the contrary situation, the input function is inactive. When the input signal with “\*” sign is through on to +24V, the input function is inactive. In the contrary situation, the input function is active.

**Note 4:** The effectiveness of +24V, COM terminals are equivalent to the CNC power box terminals that have the same names.

## 3.2 Input Signal

### 1) DC input signal A

DC input signal A is the signal from keys, limit switch, contacts of the relay (including DECX, DECZ, ESP, TCP, ST, SP, DITW, X16, etc.) at the machine side.

a) The contact point at the machine side should meet the following requirements:

Contact capacity: DC30V, 16mA above

Leakage current between contacts in open circuit: 1mA bellow (voltage 26.4V)

Voltage drop between contacts in closed circuit: 24V bellow (current 8.5mA, including cable voltage drop)

b) Signal loop of this kind of signal is shown in Fig. 3-2-1

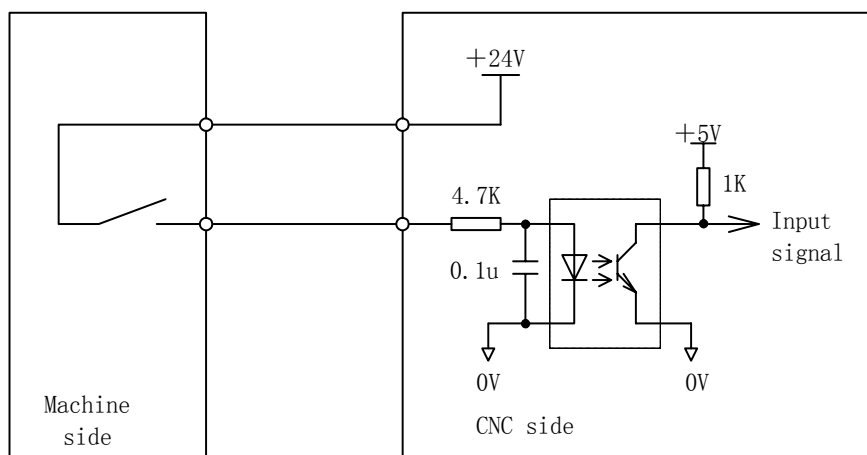


Fig. 3-2-1

## 2) DC input signal B

DC input signal B is the signal from the machine tool to the CNC, which is used at high speed (T1~T8)

## a) The contact point at the machine side should meet the following requirements:

Capacity of the contact: DC30V, 16mA above

Leakage current between contacts in open circuit: 1mA bellow (voltage 26.4V)

Voltage drop between contacts in closed circuit: 24V bellow (current 8.5mA, including cable voltage drop)

## b) Signal loop of this kind of signal is shown in Fig. 3-2-2

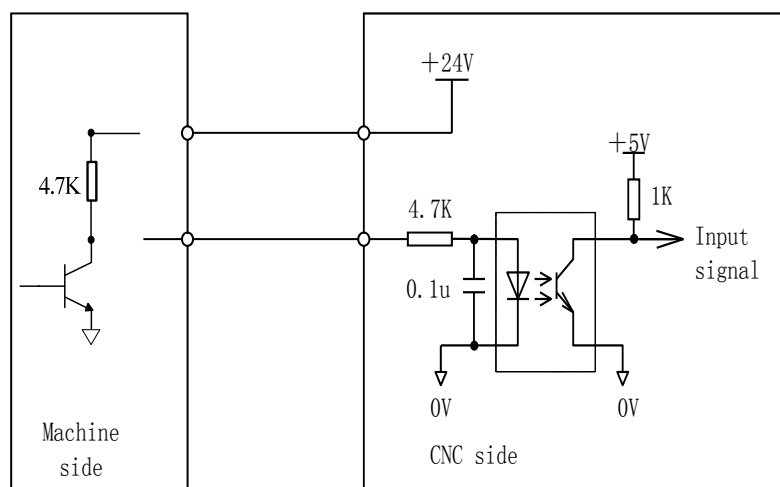


Fig. 3-2-2

## 3.3 Signal Expalnation

## 1) Input signal

## a) T01~T08 tool number signal

Effective level is low level. When one of the signals is low level signal, it indicates that the tool post is at this tool number position. Connection is shown as the following figure, 8 external pull-up resistors are connected.

By setting the parameter NO:9#1, it is able to set tool post in position level signal (0: high level is effective 1: low level is effective).

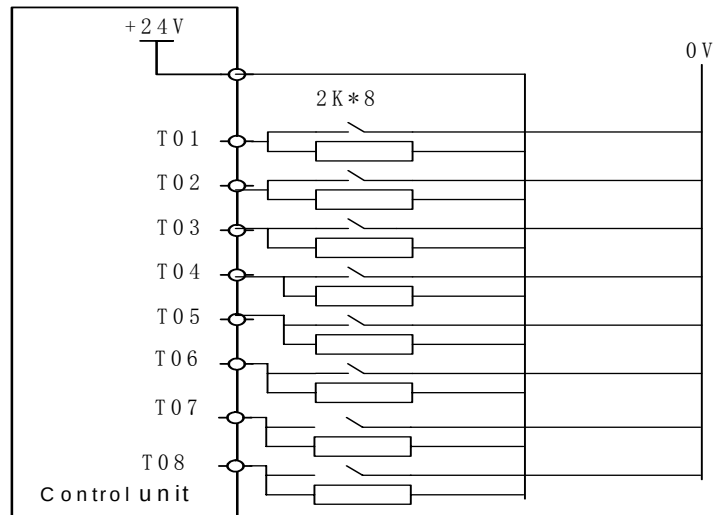


Fig. 3-3-1

## b) TCP: Tool post clamping signal

In the process of tool changing, the system outputs tool post CW signal (**TL-**) when the tool post is in position and then begins to check the clamping signal. After receiving this signal, the parameter P85 setting time is delayed, and the tool post CW signal (**TL-**) is cut off, T code command is finished, and the next block is executed. After the system outputs the tool post CW signal, if the system does not receive **TCP** in the time set by the parameter **P88**, the system will issue alarm and cut off the tool CW signal. If tool post clamping signal is not output from the tool post controller, set parameter NO.9#0 TCPS to 0 and the tool post clamp signal is not necessary to be connected.

Current tool number is stored at diagnosis **P49**. The system modifies this value after tool changing is finished normally. The values at the right corner are the same. After T command is specified, for some reason these two values are not the same. T displays the number of the specified tool, while P49 keeps the tool number before tool changing. When the number of specified tool is consistent with that of P49, tool changing is not performed. Therefore, when the current tool number is not identical with the one of P49, tool changing should be performed until they are the same.

In manual tool changing, T code and P49 are changed into new value after the tool changing is finished.

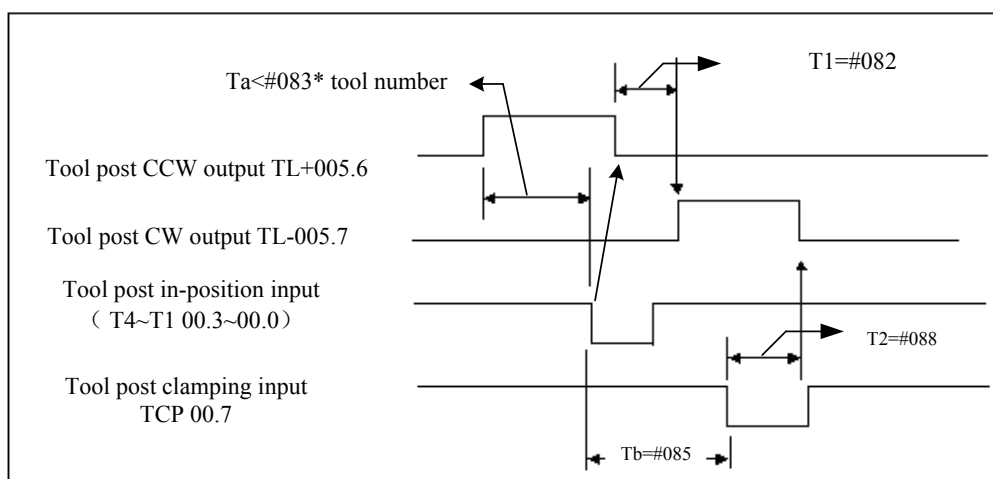
**Tool changing sequence diagram:**

Fig. 3-3-2

**#83** In the diagram is time parameter set by the corresponding parameter number.

When  $Ta \geq \#083 \times \text{currently changed tool number}$ , alarm 185 issues: tool changing time is too long. If CCW rotation from the tool post for  $Ta$  (time), the tool position signal is still not received, alarm occurs.

When  $Tb \geq \#088$ , alarm 186 occurs: Tool post CW rotation locking signal is not received in the time of tool post CW rotation locking time.

By setting the parameter **NO:9#1 TSGN**, it is able to set whether high level or low level is effective for the tool post in position signal ( $T8 \sim T1$ ).

**TSGN 0:** Tool post in-position signal is high level active (normally open)

**1:** Tool post in-position signal is low level active (normally closed)

By setting the parameter **NO:9#0 TCPS**, it is able to set whether high level or low level is effective for the tool post clamping signal (\*TCP).

**TCPS 0:** Tool post clamping signal is low level active (normally open)

**1:** Tool post clamping signal is high level active (normally closed)

c) **DECX, DECZ deceleration switch signal**

The signal is a normally-closed contact, which is used in mechanical reference return. The process of the reference return is as follows:

After selecting the mechanical origin mode, then press the manual feed key of the corresponding axis, and the machine tool will move to the reference point at rapid traverse speed. When reference return deceleration signal (**DECX, DECZ**) contact is disconnected (press down the deceleration switch), the feed rate is decelerated immediately. Then the machine tool moves at the regular low speed [set by the parameter P43. When the deceleration switch is released, the deceleration signal contact is closed again. In addition, CNC begins to detect the encoder one-turn signal or magnetic switch signal (PC signal). If this signal is high level, the motion will be halted. In addition, the corresponding zero return indicator on the operation panel lights up for machine zero return completion. Manual feed is always inactive before the zero return key is released.

Reference return direction of each axis can be set by the parameter. The connection diagram is as follows:

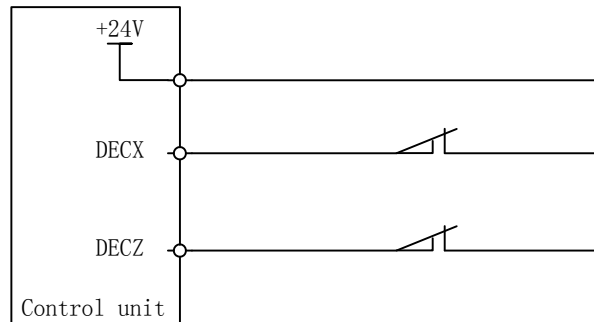


Fig. 3-3-3

d) The functions of automatic cycle start signal ST and feed pause are the same as that of the system panel. When **NO:14#0 MST=0**, **NO:14#1 M@SP=0**, signal ST and SP are active, otherwise, the signal of this point is empty. When **NO:14#1 M@SP=0**, external feed hold switch is necessary, otherwise, feed hold signal will be active all the time.

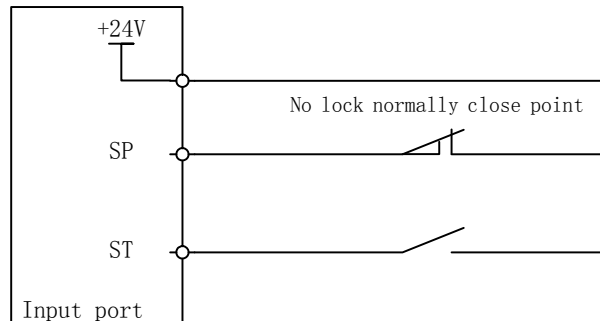


Fig. 3-3-4

e) Emergency stop signal ESP

This signal is a normally-closed signal. When the contact is disconnected, the control system resets, and the machine tool stops urgently. After the emergency stop, the system ready signal EN will be disconnected. At the same time, motion command output is blocked.

The system is designed with software limit function to perform overtravel check. It is unnecessary to fix a limit switch for overtravel checking. However, when the drive unit fault occurs, the motion of the machine tool exceeds the software limit zone. In order to stop the machine tool, a stroke limit switch is necessary. Its connection figure is as follows:

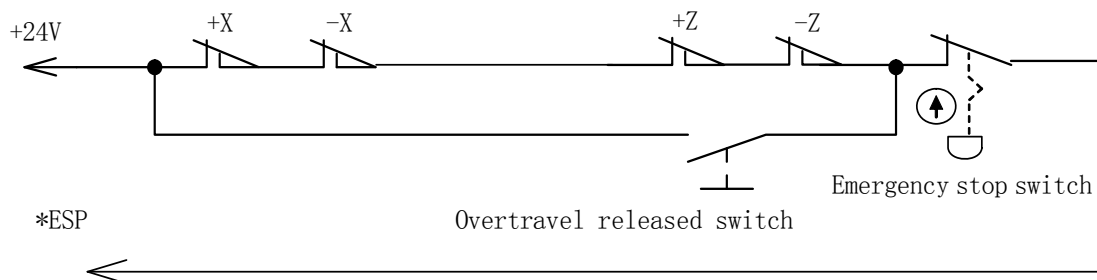


Fig. 3-3-5

When the connection in the above figure is used, if the parameter **NO:14#3 MESP=0** is set to 1, the above function is not active. The emergency signal (\*ESP) explanted here is standard connection. This connection method should be used as far as possible when users connect the signal.

## f) Hardware limit signal LIMITZ, LIMITX

This signal is a normally open signal. When the contact is disconnected, the control system resets, and the machine tool stop urgently. After the emergency stop, the system hardware limit alarm occurs. At the same time, motion command output is blocked.

When this function is active, the hardware limit of each axis can be separated from **ESP**, and limitation of each axis will be reflected correctly.

When this function is used, the parameter **NO:8#2 LMTX**. **NO:8#3 LMTZ** should be set to 1. If it is set to 0, the function is not active. Moreover, by setting **NO:8#4 LTVX**. **NO:8#5 LTVZ**, it is able to set the signal is high level active or low level active. When it is active at high level, the alarm occurs when the signal is through on to +24V. When it is active at low level, the alarm occurs when the signal is through on to 0V. When the alarm occurs, enter into the manual mode, move the overtravel axis reversely. The alarm will be released when the axis departs from the stroke switch.

g) External alarm signal **OW1**, **OW2**

This signal is normally open contact signal. When the contact is connected, the control system resets, making the machine tool stop urgently. After the emergency stop, the external alarm occurs in the system. At the same time, motion command output is blocked.

This signal is an optional one, which can be used as required.

When this function is used, the parameter **NO:12#6 WAR1**. **NO:12#7 WAR2** should be set to 1. If it is set to 0, the function is not active. Moreover, by setting **NO:12#3 WA2**. **NO:12#2 WA1**, it is able to set the signal is high level active or low level active. When it is active at high level, the alarm occurs when the signal is through on to +24V. When it is active at low level, the alarm occurs when the signal is through on to 0V. When the alarm occurs, operation can be performed after reset.

h) **SPEN**, **SAR feed and spindle enable input signal**

These two signals are normally open signals. When SAR contact is closed, spindle can be started and the cycle start can be performed. When SPEN contact is closed, spindle can be started while the cycle start can not be performed. If these two contacts are disconnected, the spindle can not be started, and the cycle start can not be performed either.

When the spindle is rotating, if disconnect both signals, the spindle will stop immediately. After one contact is connected, the CNC starts the spindle automatically, restoring the previous speed.

If the SPEN signal is disconnected in program execution, the CNC in feed hold status. After SAR signal is closed, the current program can be performed by pressing cycle start key.

The function is active when the parameter **NO:15#1 SPEN** is set to 1. If it is set to 0, the function is not active.

i) Safety door detection signal **DOOR**

This signal is a normally closed signal. The system resets when the contact is disconnected, which makes the machine tool stop urgently. After the emergency stop, the alarm that the safety door is not closed occurs in the CNC. At the same time, motion command output is blocked.

This function is active when the parameter **NO:16#5 SFDR** is set to 1. The CNC can not run if the safety door is open, the door should be closed in operation.

When the parameter is set to 0, CNC does not check whether the safety door is closed or not. By setting parameter **NO:16#6 SFDV**, it is able to set the signal is high level active or low level active. When it is high level active, the safety door is closed when the signal connects to +24V. However, the alarm is issued when the signal is disconnected or it connects to 0V. When it is low level

active, the safety door is closed when the signal connects to 0V. However, the alarm is issued when the signal connects to +24.

This function is active in Auto mode, safety door is not detected in other modes.

j) PCH Pressure check signal

This signal is normally closed contact signal. The system resets when the contact is disconnected, which makes the machine tool stop urgently. After the emergency stop, low pressure alarm occurs in the CNC. At the same time, motion command output is blocked.

This function is active when the parameter **NO:16#4 PSCK** is set to 1. The CNC can not be started normally if the pressure is low. By setting the parameter **NO:16#3 PSCV**, it is able to set the signal is high level active or low level active. When it is high level active, the signal is through on to +24V, which indicates the pressure reaches the specified value. If the signal is disconnected with +24V or connected to 0V, the alarm is issued. When it is low level active, the signal is through on to 0V, which indicates the pressure reaches the specified value. The alarm is issued when the signal is through on to +24V.

When this function is active, once the low pressure alarm is detected and the signal hold time exceeds the value set by the parameter P92, the alarm is issued in CNC. At this time, the axis feed and spindle stop, automatic cycle can not be started, cooling and lubricant are cut off. The alarm can be released by resetting or powering off.

k) GR1, GR2 Gear in-position signal

This signal is normally open contact signal. See 3.5 for the functions.

l) Expansion user input signal

The system added two user defined input signals.

Code format: **M91**

**M92**

**M93**

**M94**

Input points correspond to the codes **M91, M92, M93, M94** are not defined. User can define it as required.

**M91, M92** correspond to user defined input point 1. **M93, M94** correspond to user defined input point 2. If status of the input points conform to the one specified by the codes, the next block is executed sequentially. If not, it will wait.

**M91** code detects input status of the user 1. It waits (input terminal connects with 0V) until input is inactive.

**M92** code detects input status of the user 1. It waits (input terminal disconnects with 0V) until input is active.

**M93** code detects input status of the user 2. It waits (input terminal connects with 0V) until input is inactive.

**M94** code detects input status of the user 2. It waits (input terminal disconnects with 0V) until input is active.

### 3.4 Output Signal

1) Output signal

DC output signal is used for driving the relay and indicator at the machine tool side. Transistor is used

in drive circuit, which includes: **S1~S4, M3, M4, M5, M8, DOTWJ (M10), DOTWS (M11), DOQPJ (M12), DOQPS (M13), M32, TL-, TL+, SPZD, M51~M70.**

a) Specifications of the transistor used for outputting

- ① Maximum load current when output is ON: below 200mA including transient current.
- ② Saturation voltage when output is ON: Max. value is 1.6V when the current is 200mA. Typical value is 1V.
- ③ Withstand voltage when output is OFF: below 24+20% including transient current.
- ④ Leakage current when output is OFF: below 100 $\mu$ A.

b) Output circuit

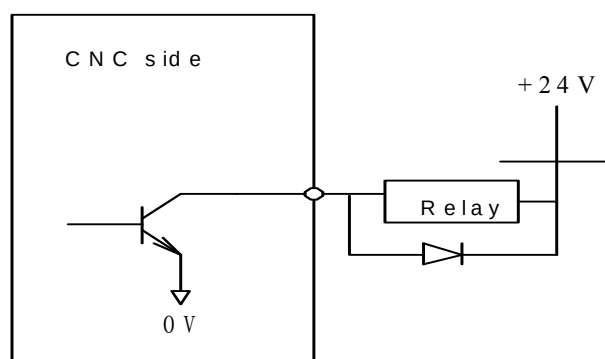


Fig. 3-4-1

All output signals for this system are provided by Darlington tube. Corresponding Darlington tube is connected when output is active. Except for pulse signals (non output hold) TL-, TL+, SPZD, others are level signals (output hold), the common port of the signal is +24V.

2) Explanation of output signal

a) S1~S4 Spindle speed switching value control signal

Share an output port with M41~M44. When spindle gear control is selected, S1, S2 (second gear) or S1~S4 (fourth gear) functions are active.

Only one signal in S1~S4 is active.

b) TL+, TL- tool change signal

TL+ is CCW tool change signal, TL- is CW tool change signal.

c) M03, M04, M05 are spindle control M code.

M03 is for spindle CCW rotation, M04 is for spindle CW rotation, M05 is for spindle stop.

d) M08, M09 (no internal signal output) cooling control.

M08 is for cooling ON. M09 is for cooling OFF.

e) DOTWJ, DOTWS are tailstock control switches. DOTWJ: tailstock advance. DOTWS: tailstock retreat. See 3.7 for details.

f) DOQPJ, DOQPS are for chuck clamp control. DOQPJ is for chuck clamp. DOQPS is for chuck unclamp.

DOQPJ shares an output port with Y16 and DOQPS. DOQPJ is for chuck clamp. DOQPS is for chuck unclamp. The function is active when automatic chuck control is selected. It is reverse in outer chuck mode. See 3.6 for details.

g) M32, M33 (no internal signal output) lubricant control

M32 is for lubricant ON. M33 is for lubricant OFF. When **NO:13#2 JLB** is set to 0, oil is supplied continuously for the system. If it is set to 1, oil is supplied intermittently for the system. Oil supply time is set by the parameter **P107**. Intermittence time is set by the parameter **P108**.

h) SPZD Spindle brake signal SPZD

Action relationships are as follows:

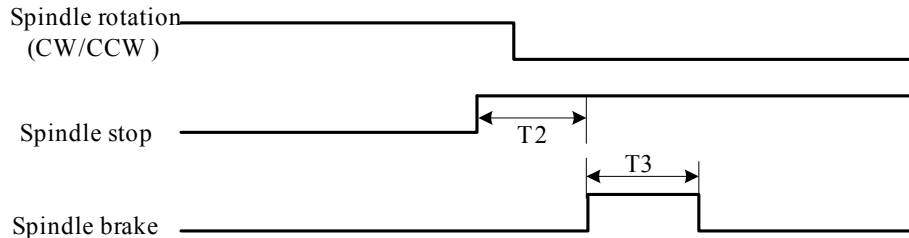


Fig. 3-4-2

T2 is delay time from outputting spindle stop signal to outputting spindle break signal. It is set on the parameter number P89.

T3 is the holding time of the spindle brake, which is set on the parameter P90.

i) STAR/TRIAN Star triangle start signal STAR/TRIAN

It is active when parameter **NO:15#7 STAR** is set to 1. **STAR** signal output is active after the spindle is started. The time set by the parameter **P93** is delayed, signal **STAR** is off and **TRIAN** signal is output.

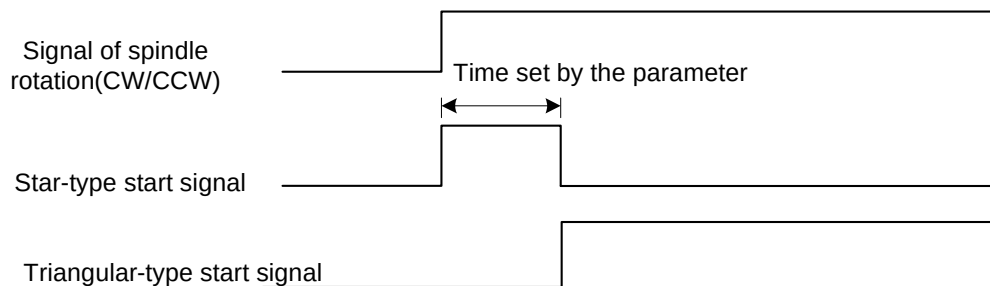


Fig. 3-4-3

j) WARN Output alarm signal WARN

It is active when the parameter **NO:15#6 OWAR** is set to 1. The signal is active if alarm is issued in the CNC. After the alarm is cancelled, this signal is inactive. The signal is used for warning customers that CNC is in alarm state.

k) Processing completion signal M30

It is active when the parameter **NO:15#5 OM30** is set to 1. After the system is powered on or M30 is executed, the system is in work completion state. At this time, the signal is active. When the CNC is processing automatically or it is running, the signal is inactive. This signal is used for warning customers that CNC is in processing completion state or in standby state.

l) Auto run signal STM

It is active when the parameter **NO:15#4 OMST** is set to 1. When the system runs automatically, the signal outputting is active. It is inactive after M30 is executed or after reset.

**WARN**, **M30** and **OSTM** are interlocked with each other. They respectively indicate alarm, standby and processing state. These signals are mainly used for controlling of the machine tool state indicators.

m) TLOCK Tool post clamp signal

The signal is active when Taiwan LIO SHING tool post is used. Please refer to the appendix 6 for details.

n) Expansion M code

The system is added 10 user defined output ports that controlled by **M51~M70** respectively. Odd commands make the signal active, and even number of commands make the signal inactive. Long signal is output in execution. The specific definitions are as follows:

- M51 code makes output of the No. 1 user output point active (output low level)
- M52 code makes output of the No. 1 user output point inactive (output high level)
- M53 code makes output of the No. 2 user output point active (output low level)
- M54 code makes output of the No. 2 user output point inactive (output high level)
- M55 code makes output of the No. 3 user output point active (output low level)
- M56 code makes output of the No. 3 user output point inactive (output high level)
- M57 code makes output of the No. 4 user output point active (output low level)
- M58 code makes output of the No. 4 user output point inactive (output high level)
- M59 code makes output of the No. 5 user output point active (output low level)
- M60 code makes output of the No. 5 user output point inactive (output high level)
- M61 code makes output of the No. 6 user output point active (output low level)
- M62 code makes output of the No. 6 user output point inactive (output high level)
- M63 code makes output of the No. 7 user output point active (output low level)
- M64 code makes output of the No. 7 user output point inactive (output high level)
- M65 code makes output of the No. 8 user output point active (output low level)
- M66 code makes output of the No. 8 user output point inactive (output high level)
- M67 code makes output of the No. 9 user output point active (output low level)
- M68 code makes output of the No. 9 user output point inactive (output high level)
- M69 code makes output of the No. 10 user output point active (output low level)
- M70 code makes output of the No. 10 user output point inactive (output high level)

### 3.5 Spindle Automatic Gearing Control

When spindle frequency conversion is selected (**0~10V** analog voltage output), automatic spindle mechanical gear shifting is available for 4 gears. By setting parameter P55~P58 based on gears, making the speed of S code is consistent with the actual speed.

Relevant parameter: bit parameter **NO:01#4:**

|   |   |   |  |  |  |             |  |  |  |  |
|---|---|---|--|--|--|-------------|--|--|--|--|
| 0 | 0 | 1 |  |  |  | <b>MDSP</b> |  |  |  |  |
|---|---|---|--|--|--|-------------|--|--|--|--|

- MDSP** =1: Spindle is analog volume control.
- =0: Spindle is other mode.

Bit parameter P **NO:10#3. 4. 6. 7:**

|   |   |   |             |             |  |             |            |            |                 |            |
|---|---|---|-------------|-------------|--|-------------|------------|------------|-----------------|------------|
| 0 | 1 | 0 | <b>AGER</b> | <b>AGIN</b> |  | <b>AGIM</b> | <b>SPT</b> | <b>TRD</b> | <b>G0&amp;T</b> | <b>TSS</b> |
|---|---|---|-------------|-------------|--|-------------|------------|------------|-----------------|------------|

- AGER** =0: Spindle automatic gearing is inactive.
- =1: Spindle automatic gearing is active.
- AGIN** =0: Not detect in-position signal when shifting to gear 1,2.
- =1: Detect in-position signal when shifting to gear 1,2.
- AGIM** =1: Gear in-position signal is high level active.

- =0: Gear in-position signal is low level active.  
**SPT** =1: Spindle jog in auto mode is active.  
 =0: Spindle jog in auto mode is inactive.

Input/output signal:

**GR1:** Gear 1 in-position signal

**GR2:** Gear 2 in-position signal

Note 1: There are no gear 3, 4 in-position signals.

**M41~M44:** 1~4 gear shifting output (Share a common port with S1~S4)

- ① Check whether the **AGER** of the parameter is 1 and analog spindle function is selected, otherwise, alarm 180 (M41~M44 can not be used when spindle is in gear control or spindle automatic gearing control is inactive) is issued.
- ② Check whether it is consistent with the current gear (check output state). No gearing is done if the specified gear is consistent with the current gear. If not, gear shifting will be performed based on the process ③.
- ③ Set the spindle speed to the one specified by the parameter 066. If there is motion, stop it temporarily.
- ④ After a delay (gear shifting time 1) by the parameter 063, turn off the original gear output signal and output the new gearing signal.
- ⑤ If the gear is 1 or 2, and the parameter 010 **AGIN** is 1, it jumps to ⑥, or else it jumps to ⑦.
- ⑥ Check the gear in-position input signal (By setting parameter 010 **AGIM**, it is able to set level is active at 0 or 1) of gear 1 or 2, it jumps to ⑦ if the gear in-position is done.
- ⑦ After a delay (gear shifting time 2) by the parameter 064, output spindle analog voltage by the current gear according to a value set by parameter P055~P058 (correspond to gear1~4) and finish the gearing.

**Note 2:** M40 cancels M41~M44 function output. S0 cancels S1~S4 function output.

**Note3:** Gearing signal M41~M44 is not output when power on. Default gear is 0.

### 3.6 Chuck Control (Chuck Detection Function)

1. Relevant signal: bit parameter **NO:15#3; NO:16#0. 1. 7:**

|   |   |   |             |  |  |  |             |             |             |
|---|---|---|-------------|--|--|--|-------------|-------------|-------------|
| 0 | 1 | 5 |             |  |  |  | <b>KPDW</b> |             |             |
| 0 | 1 | 6 | <b>NWKP</b> |  |  |  |             | <b>SLSP</b> | <b>SLQP</b> |

- KPDW** =1: Check chuck in-position signal.  
 =0: Not check chuck in-position signal.
- NWKP** =1: Outer chuck mode.  
 =0: Inner chuck mode.
- SLSP** =1: Check whether the chuck is clamped when the spindle is started  
 =0: Not check the chuck state when the spindle is started
- SLQP** =1: Chuck function is active  
 =0: Chuck function is inactive

2. Input/output signal:

**NQPJ:** Inner chuck clamping in-position/outer chuck unclamping in-position signal

- WQPJ:** Inner chuck unclamping in-position/outer chuck clamping in-position signal  
**DIQP:** Chuck clamping/unclamping input. It is usually controlled by the foot-switch  
**DOQPS** Chuck unclamping output  
**DOQPJ:** Chuck clamping output

3. Action time sequence:

- 1) When **SLQP=1, SLSP=0 and KPDW=0**, it only controls the clamp or unclamp of the chuck, but not decides whether the chuck clamping in position or not.
- 2) When **SLQP=1, SLSP=1 and KPDW=0**, it can not be started if the chuck is not clamped. (Alarm 044, start the spindle before the chuck is clamped)

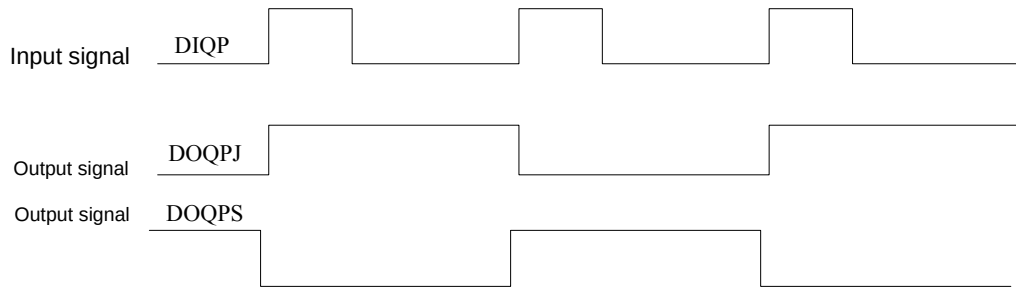


Fig. 3-6-1

**DOQPS:** Chuck unclamping output; **DOQPJ:** Chuck clamping output  
 DOQPJ and DOQPS output high resistance at power on. When CNC detects that the chuck input signal DIQP is active for the first time. DOQPJ (pin 4 of XS39) outputs 0V, chuck is clamped. After M12 is executed, DOQPS (pin 10 of XS39) outputs high resistance. DOQPJ (pin 4 of XS39) outputs 0V, and chuck is clamped. After M13 is executed, DOQPJ (pin 4 of XS39) outputs high resistance. DOQPS (pin 10 of XS39) outputs 0V, and the chuck is unclamped.

**Note:** CNC judges whether M12 is input. If M12 is not input, the system alarms when the spindle is started.

- 3) When **SLQP=1, SLSP=1, NWKP=1, KPDW=1**, CNC chooses outer chuck mode, and chuck in-position signal detecting is active:  
**DOQPS:** chuck clamping output, **WQPJ:** clamping in-position signal  
**DOQPJ:** chuck unclamping output, **NQPJ:** unclamping in-position signal  
 DOQPJ and DOQPS output high resistance at power on. When CNC detects that the chuck input signal DIQP is active for the first time. DOQPS outputs 0V, chuck is clamped. After M12 is executed, DOQPS outputs 0V. DOQPJ outputs high resistance, chuck is clamped and CNC waits for WQP signal to be in-position.. After M13 is executed, DOQPJ outputs 0V. DOQPS outputs high resistance, the chuck is unclamped and CNC waits for NQPJ signal to be in-position. As the second chuck input is active, DOQPS outputs 0V, chuck is unclamped. The chuck clamping/unclamping signal is output alternatively, i.e. the output state is changed each chuck input signal is active.

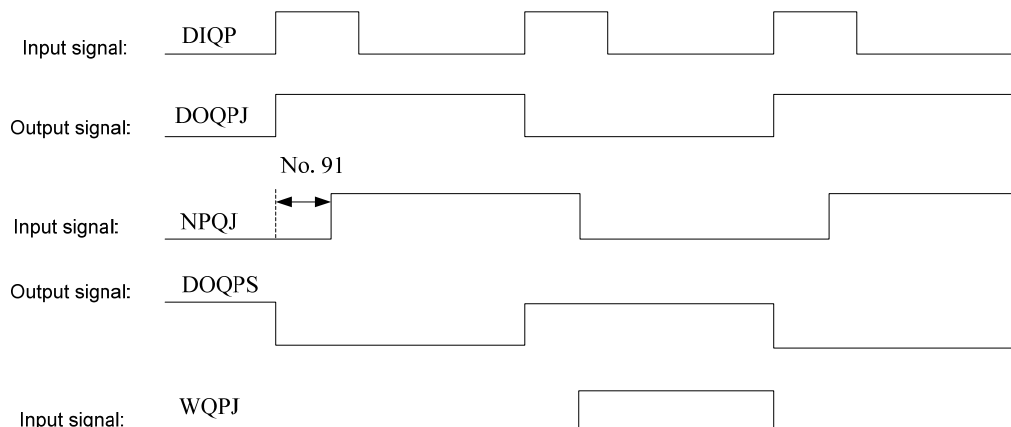


Fig. 3-6-2

**Note:** CNC detects clamping in-position signal altered it is clamped. If there is no clamp signal input, CNC alarms.

4) The interlock of the chuck and the spindle:

When SLQP=1, SLSP=1, M3 or M4 is active, alarm is issued if M13 is executed and the output is unchanged.

When SLQP=1, SLSP=1, KPDW=1, if M12 is executed in MDI or Auto mode, CNC does not execute next code till it detects the chuck clamping in-position signal is active. When the chuck input signal DIQP is active in Manual mode, the panel spindle CCW, CW keys are inactive till it detects the chuck clamping in-position signal is active. In spindle running or auto cycle processing, DIQP input signal is inactive. DOQPS, DOQPJ are held on at CNC reset and emergency stop.

When SLSP=0, KPDW=1, it does not detect whether the chuck is clamped or not. However, when chuck clamp/unclamp in-position command is executed, action is finished till in-position signal is detected. (It must be careful to use this mode)

**Note 1:** DOQPJ and DOQPS output 0. DOQPJ outputs 1 when chuck input signal is active for the 1<sup>st</sup> time.

**Note 2:** DIQP is inactive in spindle running or auto cycle processing.

### 3.7 Tailstock Control (Tailstock hierarchical advance/retrac control)

1. Relevant parameter: bit parameter **NO:15#3:** **NO:16#2:**

|   |   |   |  |  |  |  |            |             |  |  |  |
|---|---|---|--|--|--|--|------------|-------------|--|--|--|
| 0 | 1 | 5 |  |  |  |  | <b>WZ2</b> |             |  |  |  |
| 0 | 1 | 6 |  |  |  |  |            | <b>SLTW</b> |  |  |  |

**SLTW** =1: Tailstock function is active

=0: Tailstock function is inactive

**WZ2** =0: Tailstock hierarchical advancing/retracing function is inactive

=1: Tailstock hierarchical advancing/retracing function is active

2. Input/output signal:

**DITW:** Tailstock clamping/unclamping input

**DOTWJ:** Tailstock clamping output signal

**DOTWS:** Tailstock unclamping output signal

**DOTWJM:** Tailstock slow advancing output signal

**DOTWJJ:** Tailstock lock output signal

**WZ1J:** Tailstock slow advancing detection input signal

**WZ2J:** Tailstock in-position detection input signal

3. Action time sequence:

1) When SLTW=1, WZ2=0,

DITW: Tailstock clamping/unclamping input

M10/M11 is tailstock clamping/unclamping control signal respectively.

After M10 is executed, DOTWS outputs high resistance, DOTWJ outputs 0V, and tailstock advances.

After M11 is executed, DOTWJ outputs high resistance, DOTWS outputs 0V, and tailstock retreats.

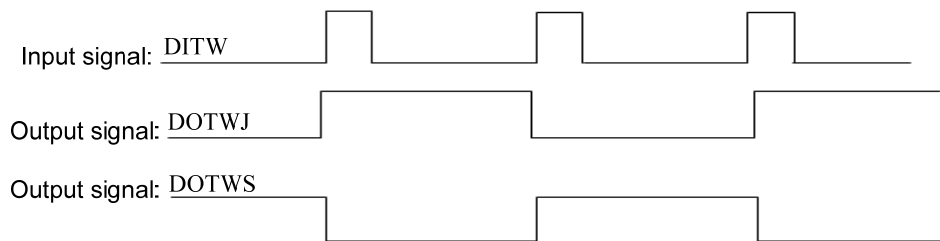


Fig. 3-7-1

2) When SLTW=1, WZ2=1, S3/S4/M43/M44 function is inactive. If the command is executed, CNC will issue alarm: illegal command. Maximum tools number is 4.

After M10 is executed, CNC DOTWS outputs high resistance, DOTWJ outputs 0V, and tailstock advances rapidly. When WZ1J signal is active, DOTWJ outputs high resistance, DOTWJM outputs 0V, tailstock advances slowly. When WZ2J signal is active, DOTWJM outputs high resistance, DOTWJJ outputs 0V, tailstock is locked and tailstock advancing is finished.

After M11 is executed, DOTWJJ outputs high resistance, tailstock is unclamped. DOTWS outputs 0V, tailstock retreats rapidly, and the tailstock retreat is finished.

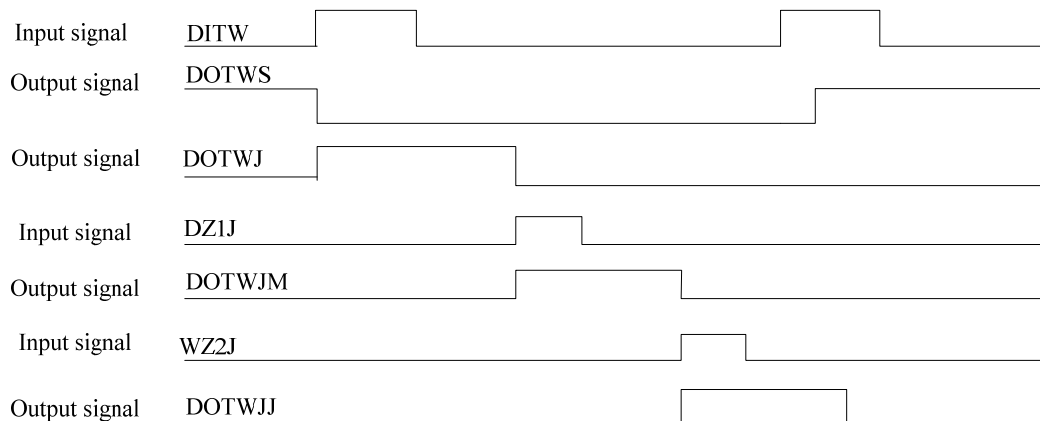


Fig. 3-7-2

**Note 1:** DOTWJ and DOTWS output 0 when power on. DOTWJ outputs 1 when chuck input signal is active for the 1<sup>st</sup> time.

**Note 2:** DITW input is inactive in spindle running or auto cycle processing.

### 3.8 External Program Lock

External program lock is an external key switch, which is used to prevent work programs from altering.

Relevant: bit parameter **NO:13#6. #7:**

|   |   |   |       |      |  |  |  |  |  |  |  |
|---|---|---|-------|------|--|--|--|--|--|--|--|
| 0 | 1 | 3 | PLCKV | PLCK |  |  |  |  |  |  |  |
|---|---|---|-------|------|--|--|--|--|--|--|--|

**PLCKV** =1: Program lock is high level active

=0: Program lock is low level active

**PLCK** =1: Program lock function is active

=0: Program lock function is inactive

Input/output signal:

**PROLOCK:** External program lock

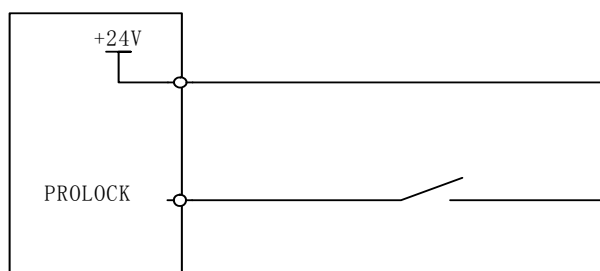


Fig. 3-9-1

### 3.9 External Feed Pause and Spindle Pause

External feed pause and spindle pause is a three-phase code-chosen-switch. They are used to stop program in auto running or stop the spindle. When the three-phase is on the normal position, the modes “press reset key again” or “execute program directly” can be selected.

Relevant parameter: bit parameter **NO:15#0. #1:**

|   |   |   |  |  |  |  |  |  |      |       |
|---|---|---|--|--|--|--|--|--|------|-------|
| 0 | 1 | 5 |  |  |  |  |  |  | SPEN | SPENC |
|---|---|---|--|--|--|--|--|--|------|-------|

**SPEN** =1: Feed and spindle enable input are active

=0: Feed and spindle enable input are inactive

**SPENC** =1: Feed and spindle control

=0: Spindle control

**Note:** When SPENC and SPEN are set to 0, spindle enable function is inactive. When SPENC is set to 0, SPEN is set to 1, spindle enable function is active. When SPEN signal of the input port is high level, the spindle can be started by the CNC. When SPENC and SPEN are set to 1, SAR of the input port is high level input, the spindle can be started by the CNC and feed is enabled. When SPEN signal is high level, spindle is enabled, while feed is forbidden. When SAR and SPEN are not connected to +24V, spindle rotation and feed are forbidden.

Input/ output signal:

**SAR:** External feed hold

**SPEN:** External spindle hold

This function has three connection states:

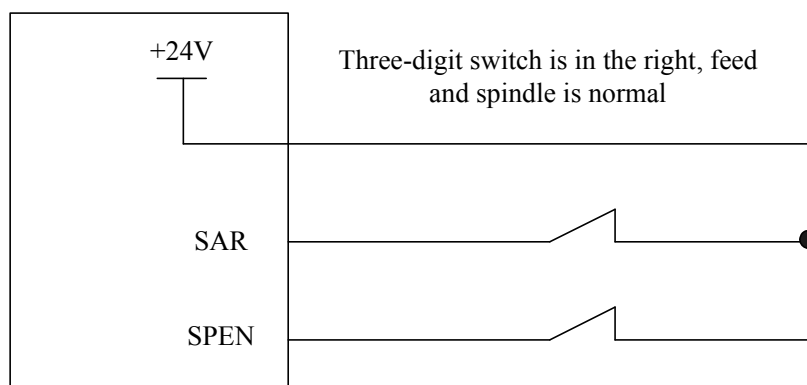


Fig.3-9-1

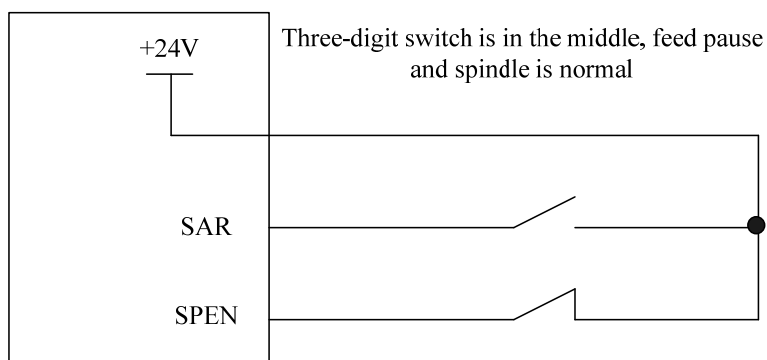


Fig.3-9-2

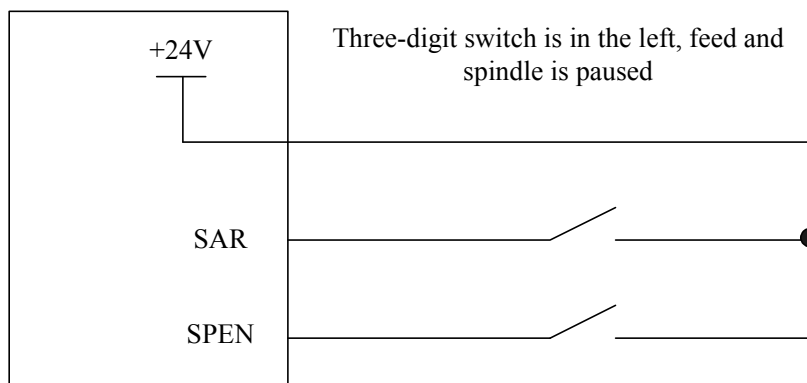


Fig.3-9-3

### 3.10 I/O Signal Diagnosis List

All I/O signals of the system are described in this diagnosis list.

#### 1) List of the signals from machine tool to CNC

| No.                 | 7        | 6        | 5        | 4        | 3       | 2       | 1       | 0        |
|---------------------|----------|----------|----------|----------|---------|---------|---------|----------|
| 001                 | X17      | X16      | X15      | X14      | X13     | X12     | X11     | X10      |
|                     | TCP      | DIQP     | *DECX    | DITW     | *SP     | ST      | DECZ    | *ESP     |
| Socket<br>pin No. → | XS39: 12 | XS39: 11 | XS40: 1  | XS40: 2  | XS40: 7 | XS40: 8 | XS40: 9 | XS40: 10 |
| 002                 | T8       | T7       | T6       | T5       | T4      | T3      | T2      | T1       |
|                     | XS40: 19 | XS40: 20 | XS40: 21 | XS40: 22 | XS40: 3 | XS40: 4 | XS40: 5 | XS40: 6  |
| 003                 | SAR      | LTZ      | LTX      | SPEN     | PSCK    | DOOR    | GR2     | GR1      |
|                     |          |          |          |          |         |         |         |          |
| 004                 |          |          |          |          |         | LCK     | OWA2    | OWA1     |
|                     |          |          |          |          |         |         |         |          |

## 2) List of the signals from CNC to machine tool

| No.:                | 7       | 6       | 5        | 4       | 3       | 2       | 1       | 0       |
|---------------------|---------|---------|----------|---------|---------|---------|---------|---------|
| 005                 | Y17     | Y16     | Y15      | Y14     | Y13     | Y12     | Y11     | Y10     |
| Diagnosis No.       | SPZD    | M12     | M05      | M32     | M08     | M10     | M04     | M03     |
| Socket<br>pin No. → | XS39:17 | XS39: 4 | XS39:16  | XS39: 6 | XS39:15 | XS39: 2 | XS39:3  | XS39: 7 |
| 006                 | X27     | X26     | X25      | X24     | X23     | X22     | X21     | X20     |
|                     | TL-     | TL+     | M13      | M11     | S04/M44 | S03/M43 | S02/M42 | S01/M41 |
| Socket<br>pin No. → | XS40:13 | XS40:12 | XS39: 10 | XS39: 9 | XS39: 8 | XS39:14 | XS39: 1 | XS39: 5 |

## 3.11 Function Description

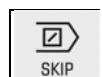
In diagnosis page, 【machine panel】 interface is shown as figure 3-12-1:

|                                                                                              |             |                 |       |
|----------------------------------------------------------------------------------------------|-------------|-----------------|-------|
|  OPERATOR | 00001 N0000 | Message         | 10:13 |
| MLK (KEY1) : *OFF ON                                                                         |             | Axis Rev        | 0000  |
| AFL (KEY2) : *OFF ON                                                                         |             | Tool No         | 0100  |
| SBK (KEY3) : *OFF ON                                                                         |             | Fact Rate       | 0     |
| DRN (KEY4) : *OFF ON                                                                         |             | Cool Mes        | OFF   |
| BDT (KEY5) : *OFF ON                                                                         |             | Axis Mes        | Stop  |
| POSITION (RELATIVE)                                                                          |             | G00 G98 G97 G40 |       |
| U 0.000                                                                                      |             |                 |       |
| W 0.000                                                                                      |             |                 |       |
|                                                                                              |             | MDI             |       |

Fig. 3-12-1

On machine panel page, 1, 2, 3, 4, 5 keys are used to shift machine lock, M.S.T. lock key, single block, dry

run, block skip switch. Their functions are the same as that of the keys



## Chapter Four Machine Tool Debugging

The debugging and trial run methods and steps are described in this chapter. The corresponding operations can be performed after debugging by the following steps.

### 4.1 Debugging Preparation

Debug the system according to the following steps:

- CNC connection: correct connection is the foundation on which debugging is performed smoothly;
- Drive unit parameter setting: set motor type parameter, control mode setting;
- CNC parameter setting: control parameter, speed parameter setting.

### 4.2 System Power On

- Switch on air break switch in the electric cabinet.
- Connect to the air break controlling DC 24V or fuse, and check whether DC24V power is normal.
- Check whether the other power is normal.
- Turn on the CNC system.

### 4.3 Emergency Stop and Limit

This system has software limit function. It is suggested that hardware limit is employed by fixing the stroke limit switches at the positive or negative directions of the axis. The connection is shown as follows:

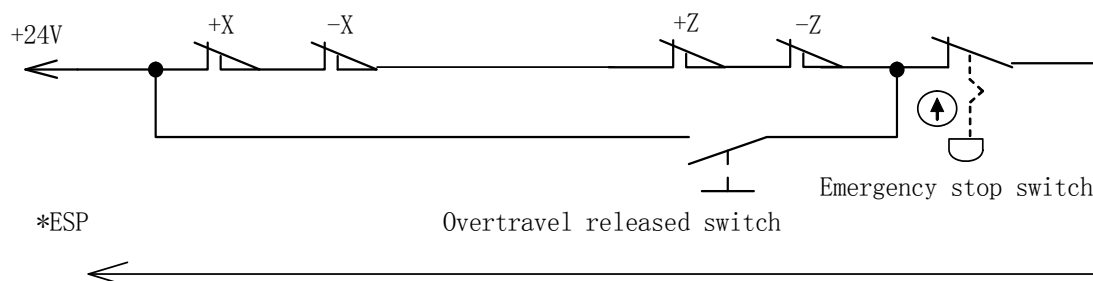


Fig. 4-3-1

Emergency stop signal can be shielded by parameter **NO:14#3 MESP**

When **MESP=1**, emergency stop is inactive. It is for debugging.

When **MESP=0**, emergency stop is active. It can be used normally.

Diagnosis **NO:01#0** can be used to check whether the emergency signal is input or not. If it is set to 1, which indicates that the emergency stop switch is connected (switching signal is normally closed signal).

In Auto mode or MPG mode, slowly move the axes to testify the activity of the stroke limit switch, correctness of the alarm display, activity of the overtravel release button. When overtravel occurs or emergency stop button is pressed, alarm 254 “the system is not ready” will be issued by CNC system. Press the overtravel release button on the panel and move the axis reversely until it detaches the stroke switch, the alarm will be eliminated.

## 4.4 Drive Axis Debugging

### 1) Drive unit alarm

If X-axis or Z-axis drive unit alarm displays when power on, user must first check whether the drive unit alarm indicator is lighted up or not (for stepping drive unit) or check whether the alarm is displayed (for servo drive unit). If alarm occurs in drive unit, process the drive unit first, and then check whether the drive alarm line is cut off or not. If none above phenomenon occurs, the reason is that the level set by alarm parameter does not match the drive unit. Therefore, alter #0 and #1 the parameter **NO:8** (**ALMX** and **ALMZ** correspond to X and Z axes respectively). After the parameter



alteration, press key to eliminate drive unit alarm.

### 2) Direction adjustment of the driving motor

If operation direction of the drive unit is inconsistent with coordinate direction of the computer, alter #0 and #1 the parameter **NO:7#0** (**DIRX** and **DIRZ** correspond to X and Z axes respectively) or change direction selection switch of the transformer drive unit.

## 4.5 Gear Ratio Adjustment

### 1) Formula

**CMR**: Command multiplier coefficient [Parameter **P17~18**]

**CMD**: Command frequency division coefficient [Parameter **P19~20**]

**a**: Stepping angle (degree)

**L**: Movement amount corresponds to the step motor rotating one turn (mm)

Example: Z axial motor is connected directly. Screw pitch of the lead screw is 6mm. X axial reduction ratio is 3/5. Screw pitch of the lead screw is 4mm. Its servo gear ratios are as follows: calculating method for X axis:  $4 \times 2500 \times 3 / (4 \times 1000 \times 5) = 3/2$  that is, command multiplier coefficient of the X axis=3, command frequency division coefficient of the X axis=2. Calculating method for Z axis:  $4 \times 2500 / (6 \times 1000) = 5/3$ , that is, command multiplier coefficient of the Z axis=5, command frequency division coefficient of the Z axis=3. This gear ratio can be set to the drive unit. When stepping drive unit is used, its stepping angle is as follows:

X axis:  $360 \times 5 / (4000 \times 3) = 0.15$ , Z axis:  $360 / 6000 = 0.06$ .

## 4.6 Rapid Traverse Rate and Linear ACC&DEC Adjustment

Parameter number **P21**, **P22** respectively set the rapid traverse rate scope (**30 mm/min~30000mm/min**) of the X and Z axes.

Parameter number **P23**, **P24** respectively set the linear ACC&DEC time constant scope (**8~4000**) of the X and Z axes in rapid traverse.

Parameter number **P25**, **P26** respectively set the S-shape time constant value setting scope (**8~4000**) of the X and Z axes in rapid traverse.

Parameter number **P28** is lower value of the ACC&DEC low speed in cutting feed (**0~500mm/min**).

Parameter number **P29**, **P30** are S acceleration/deceleration time constant in cutting feed, which scope is **8~4000**.

Parameter number **P31** are linear acceleration/deceleration time constant in cutting feed, which scope is **8~4000**.

**Note:** When stepping drive unit is used, it is suggested that ACC&DEC time constant are bigger than or equal to 300.  
When servo drive unit is used, the value can be set to 100.

#### 4.7 Upper Speed of the Cutting Feed

Parameter P27 is used to set the upper speed of the cutting feed. It must be set to 8000.

#### 4.8 Adjustment for Thread Machining

##### 1) Adjustment for thread run-out width

Parameter P68 **THDCH** is used to set thread run-out length. Chamfering width = **THDCH\*1/10\*** screw pitch.

##### 2) Threading exponential acceleration and deceleration

Parameter **P69** is linear ACC&DEC constant of two axes in threading. It is usually set to 100.

Parameter **P157** is ACC&DEC time constant of the shorter axis in thread run-out.

Parameter **P158** is maximum speed of the shorter axis in thread run-out.

##### 3) Spindle setting

Parameter P59 is for threading is performed after percentage is stable.

Parameter P59 is spindle speed sampling period (×4ms).

#### 4.9 Machine (mechanical) Zero Adjustment

Users have to make sure that zero-return deceleration switch is connected correctly. It can be checked by diagnosis NO:1#5 DECX and NO:1#1 DECZ. Make sure that zero point signal is linked correctly. It can be checked by diagnosis **NO:9#0 PCX** and **NO:9#1 PCZ**. If it is right, this digit will be changed into 1 after the motor rotates once.

Parameter **NO:6#4. 5** (**ZCX** and **ZCZ** correspond to X and Z axes respectively) are used to set the type of the zero return. They are usually set to 0. For the detailed explanations, please refer to *machine zero return*, chapter 8.

Parameter **NO:5#1. 2** (**ZMX** and **ZMZ** correspond to X and Z axes respectively) are used to set direction of zero return. They are usually set to 1, Which indicates CCW zero return.

#### 4.10 Offset between Lead Screws

Parameters P36, P37 are offset between lead screws of the X, Z axes. Thereinto, No.36 is diameter value.

Backlash compensation should be detected by dialgaug, clock gauge or laser detector, it doesn't recommend to be detected by handwheel or single-step mode, because backlash compensation only compensate accurately can improve the precision of processing, it is suggest to use the following method:

- Edit program:

```
O0001;
N10 G01 W10 F800;
N20 W15 ;
```

N30 W1 ;  
 N40 W-1 ;  
 N50 M30 。

- Backlash compensation value (No.36,37) should be set to 0 before measuring;
- Single segment to run the program, find measure basis A after twice positioning, record current data.  
 Run 1mm in the same direction, then run 1mm to point B reverse to read current data.

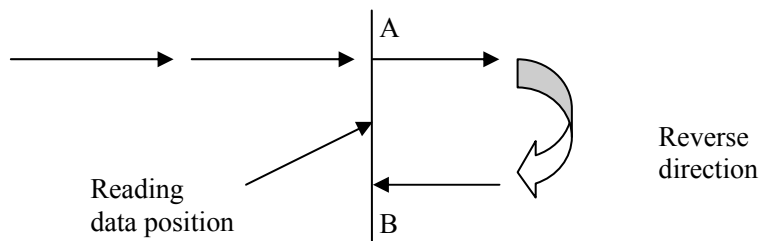


Fig.4-4 Backlash measure method drawing

- Backlash error compensation value = | data record from point A – data record from point B | ; input the count result to system parameter No.036(BKLX) or NO.037 (BKLZ).

Data A : data of dial indicator read at point A;

Data B : data of dial indicator read at point B;

pulse equivalent: 1 mu

Note 1: system parameter No.033 can set frequency of backlash compensation.

Note 2: backlash compensation should be rechecked after the machine tool has been used for 3 months.

#### 4.11 Electronic Toolpost Debugging

Make sure the cable of tool number input signal is connected correctly. It can be checked by **NO:2#1~7 (T01~T08)** . When it is right, in-position signal displays 0. Otherwise, in-position signal displays 1. (In-position signal displays 1, and it displays 0 for other situations)

Diagnosis **NO:6#6. 7 (TL-, TL+)** can be used to check whether toolpost CCW/CW rotation signals are correct in tool changing.

The following parameters should be set correctly to ensure the toolpost runs normally.

Parameter **NO:9=00001110** (**BIT0=1** indicates toolpost clamping is high level active, **BIT0=0** indicates toolpost clamping is low level active. **BIT1=1** indicates toolpost clamping is low level active, **BIT1=0** indicates toolpost clamping is high level active)

Parameter **P81=4**: Total tool number selection

Parameter **P82=10**: Tool changing time T1 (**x4ms**)

Parameter **P83=1000**: Upper time for moving a tool in tool changing

Parameter **P84=15000**: Upper time from changing the 1<sup>st</sup> tool to changing the last tool

Parameter **P85=200** : Tool post CW clamping time (**x4ms**)

Parameter **P88=500**: Alarm time of not receiving \*TCP signal

### 4.12 Spindle Adjustment

When converter controls the spindle, parameter **NO:1#4=1**. When the spindle is not controlled by converter, parameter **NO:1#4=0**.

The system could be equipped with 1024 or 1200-line encoder (outputting **A**,  $\overline{\text{A}}$ , **B**,  $\overline{\text{B}}$ , **Z**,  $\overline{\text{Z}}$  signal), which can be set by parameter P52 . When drive ratio between the spindle and encoder is not 1: 1, it can be adjusted by setting the gear number between spindle and encoder with parameter P53, P54 .

Diagnosis **NO:9#2 PCS** could be used to check screw head signal of the spindle encoder.

When the converter controls the spindle, adjust value of parameter P55 to make input rotational speed command (S command) is consistent with the actual speed.

Parameter P89 is the time from stopping spindle command to outputting spindle brake. P90 is spindle brake time. When **P89=50, P90=60**, 240ms spindle brake is performed after M05 sending 200ms.

### 4.13 Step/MPG Adjustment

When **NO:1#3=1**, MPG mode is active, Step mode is inactive. When **NO:1#3=0**, MPG mode is inactive, Step mode is active.

### 4.14 Three-state Switch Adjustment

In Auto mode, turn the knob to the right side: feed, spindle are normal; to the middle: feed stops; to the left: spindle and feed stop.

1) Three-state switch parameter setting

When parameters **NO:15#0 SPENC** and **#1 SPEN** are set to 1, three-state switch is active. When they are set to 0 at the same time, three-state switch is shielded.

2) Three-state switch signal diagnosis

Diagnosis **NO:003#7** can be used to check whether the feed pause signal is input. It indicates that three-state switch is turned to the middle if it is 0, “stop” alarm will be issued by CNC.

Diagnosis **NO:003#4** can be used to check whether the spindle pause signal is input. It indicates that three-state switch is turned to the left if it is 0, spindle stops and “stop” alarm will be issued by CNC.

3) Notice:

- a) In operation, turn the knob to the normal gear (right side), subsequent programs can be executed after “cycle start” key is pressed again.
- b) If the spindle has been started (e.g. CCW rotation), turn the knob to the left gear, the spindle will stop. Then turn the knob to the previous gear, spindle rotation state will be restored (revert to CCW rotation).
- c) Thread cutting, machine zero return, program zero return, manual, MPG are not controlled by three-state switch.

### 4.15 Other Adjustment

| Parameter No. |   |   | Diagnosis position |      |  |      |      |     |      |     |
|---------------|---|---|--------------------|------|--|------|------|-----|------|-----|
| 0             | 1 | 0 | AGER               | AGIN |  | AGIM | SPT  |     |      |     |
| 0             | 1 | 4 | KEY1               | SKEY |  |      | MESP | MOT | M@SP | MST |

|   |   |   |  |  |  |  |  |  |      |  |
|---|---|---|--|--|--|--|--|--|------|--|
| 0 | 1 | 5 |  |  |  |  |  |  | SPEN |  |
|---|---|---|--|--|--|--|--|--|------|--|

|   |   |   |  |  |  |  |  |      |      |      |
|---|---|---|--|--|--|--|--|------|------|------|
| 0 | 1 | 6 |  |  |  |  |  | SLTW | SLSP | SLQP |
|---|---|---|--|--|--|--|--|------|------|------|

**AGER=1/0:** Spindle automatic gearing active/inactive

**AGIN:** When it is 1, detect gearing in-position signal when gearing to gear 1, 2; When it is 0, gearing in-position signal is not detected

**AGIM=0/1:** M411/\*SPEN. M412/\*STEN signals active level selection (0/1 active level)

**SPT=0/1:** Spindle jog function is inactive/active [ Spindle jog dwell time can be set by parameter **P94** ]

**SLTW=0/1:** Tailstock function is inactive /active

**SLQP:** When it is 1, chuck control function is active; When it is 0, chuck control function is inactive

**SLSP:** When it is 1, check chuck clamping, if the chuck is not clamped, spindle is disabled (alarm 0015, movement stops).

When it is 0, chuck clamping is not checked.

**MST=1:** External cycle start signal (ST) is shielded, it is not cycle start switch now.

**M@SP=1:** External pause signal (SP) is shielded, it is not pause switch now.

**M@SP=0:** External pause signal is active, now external pause switch should be installed at this time, otherwise, computer displays “pause” alarm.

**MOT=1:** Not check software stroke limit.

**MESP=1:** Emergency stop signal (ESP) is shielded, emergency input is inactive.

**SKEY=1:** Program switch is shielded, edit program is forbidden.

**KEY1=1:** Program switch is on when power on.

## Chapter Five Memorizing Screw-pitch Error Compensation

Only the parameter NO:2#2 is set to 1, the memorizing screw-pitch error compensation is active.

Use methods of the memorizing screw-pitch error compensation are described as follows:

### 1. Function

The minimum movement unit could be used for axes screw-pitch.

### 2. Specification

Take the reference point as the compensation origin, and set the compensation value into the parameters according to axes compensation intervals.

a) Compensated axis: X, Z axis

b) Compensation points

360 points for each axis (X, Z axis)

c) Compensation range

$0 \sim \pm 7 \times \text{compensation override}$  (minimum movement unit) for each compensation point.

Compensation override  $\times 1$ ,  $\times 2$ ,  $\times 4$ ,  $\times 8$ ,  $\times 16$  (used for whole axis)

d) Compensation intervals

| Movement unit | Min. interval | Max. interval | Unit        |
|---------------|---------------|---------------|-------------|
| Metric        | 1000          | 9999999       | 0.001mm     |
| Inch          | 1000          | 9999999       | 0.0001 inch |

(Max. compensation range = interval  $\times$  360)

Actual compensation interval: set an appropriate value in the range above according to the maximum compensation range and mechanical travel. For linear axis, if the interval set is smaller than the min. interval, compensation is frequently unable to perform at the expected position. At this time, reduce the feedrate.

### 3. Parameter setting

Set the screw-pitch error parameter as methods described bellow in MDI mode or emergency stop state.

a) Screw-pitch error compensation override

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

|   |
|---|
| 1 |
|---|

Output the product of specified compensation value and the override (used for whole axis)

b) Screw-pitch error origin

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

|                                 |
|---------------------------------|
| X-axis screw-pitch error origin |
|---------------------------------|

|   |   |   |
|---|---|---|
| 0 | 4 | 0 |
|---|---|---|

|                                 |
|---------------------------------|
| X-axis screw-pitch error origin |
|---------------------------------|

X, Z screw-pitch error origin: This is the origin on the screw-pitch error setting list.

The value from 0 to 360 in each axis can be set according to mechanical requirements.

c) Compensation interval

|   |   |   |                                                |
|---|---|---|------------------------------------------------|
| 0 | 4 | 1 | X-axis screw-pitch error compensation interval |
| 0 | 4 | 2 | Z-axis screw-pitch error compensation interval |

X, Z: screw-pitch error compensation interval.

This is setting of screw-pitch error compensation interval. The value set should be bigger than 10. Moreover, if it is set to 0, compensation will not be performed.

#### d) Setting of compensation value

The axes screw-pitch compensation values are set according to the parameter numbers in the following table.

In screw-pitch compensation interface, input X/Z compensation from 000~255. The compensation value can be set at the range 0~±7. If exceeding this range, the input value is in active. For detailed operations, please refer to *parameter display, alteration and setting, section 3.6, operation*.

**Example:**

|   |   |   |    |
|---|---|---|----|
| 1 | 2 | 0 | -7 |
|---|---|---|----|

In the above example, -7 indicates that it is the compensation value set at point 120 of X axis.

#### 4. Parameter setting examples

##### a) Screw-pitch error origin=0, compensation interval=10.000

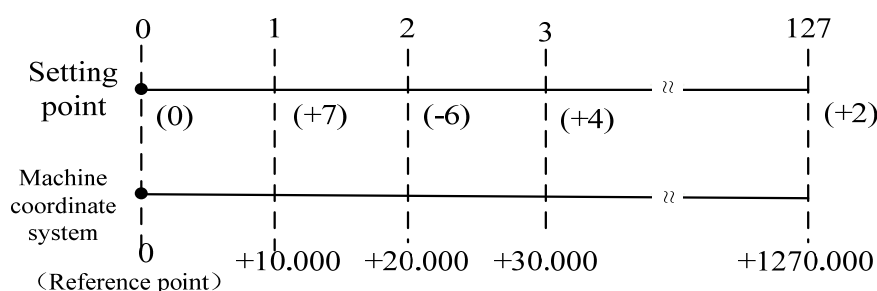


Fig. 5-1

The head of compensation point corresponds to the reference point, and compensation point1 corresponds to a point of positive 10000 moving from this reference point, and a following compensation point from this point every 10000 interval. The 127<sup>th</sup> compensation point is the compensation value at position 1270000. Therefore, at the compensation point 1, set compensation value moving from 0 to 10000, at the compensation point 2, set compensation value moving from 10000 to 20000. At compensation point N, set compensation value when moving from (N-1)×(compensation interval) to N×(compensation interval).

Above is the example for the following compensation interval error

|          |       |         |    |
|----------|-------|---------|----|
| Interval | 0     | ~ 10000 | +7 |
|          | 10000 | ~ 20000 | -6 |
|          | 20000 | ~ 30000 | +4 |

Actually the machine moves from the reference point to the point of +30000, the screw-pitch error

compensation is:

$$(+7)+(-6)+(+4)=+5$$

Machine moves from the reference point to the point of +30000, the screw-pitch error compensation is:

$$-(+4) - (-6) - (+7) = -5$$

b) Screw-pitch error origin=60, Compensation interval=10000

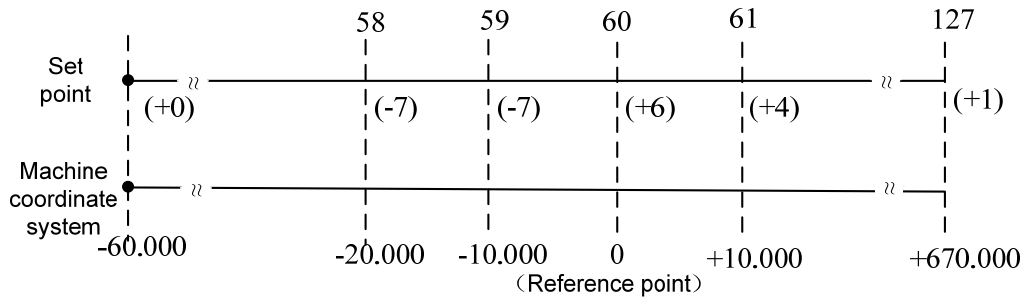


Fig. 5-2

The position No. 61, No.60 in the compensation table corresponds to the reference point, compensation point 61 corresponds to a point of positive 10000 moving from origin, and a following compensation point from this point every 10000 interval. The 127<sup>th</sup> compensation point is the compensation value at position +670000. While the position 59 corresponds to a point of negative 10000 moving from origin. Also there is a compensation point from this point every 10000 interval. The compensation point 0 corresponds to the compensation at position -600000. Therefore, the point N is set by a compensation of moving from  $(N-61) \times (\text{compensation interval})$  to  $(N-60) \times (\text{compensation interval})$ .

Above is the example for the following compensation interval error

|          |        |   |        |    |
|----------|--------|---|--------|----|
| Interval | -20000 | ~ | -10000 | -7 |
|          | -10000 | ~ | 0      | -7 |
|          | 0      | ~ | +10000 | +4 |

Actually the machine moves from -20000 to the point of +10000, the screw-pitch error compensation is:

$$(-7)+(-7) +(+4)=(-10)$$

The machine moves from +10000 to the point of -20000, the screw-pitch error compensation is:

$$-(+4)-(-7)-(-7)=(+10)$$

c) Screw-pitch error origin=127, Compensation interval=10000

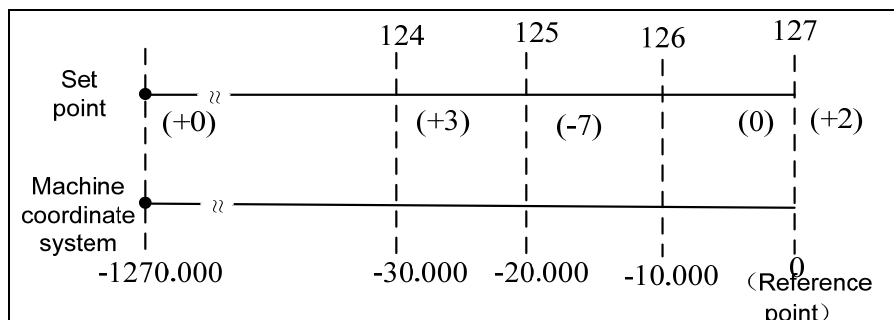


Fig. 5-3

The end of compensation table corresponds to the reference point, and compensation point 126 corresponds to a point of positive 10000 moving from this reference point, and a following compensation point from this point every -10000 interval. The compensation point 1 is the compensation value at position -1260000. Therefore, at the compensation point 127, set compensation value moving from 0 to -10000, at the compensation point 26, set compensation value moving from -10000 to -20000. At compensation point N, set compensation value when moving from  $(N-128) \times (\text{compensation interval})$  to  $(N-127) \times (\text{compensation interval})$ .

Above is the example for the following compensation interval error

|          |        |         |    |
|----------|--------|---------|----|
| Interval | -30000 | ~-20000 | +3 |
|          | -20000 | ~-10000 | -7 |
|          | -10000 | ~-00000 | 0  |

Actually the machine moves from -30000 to the reference point, the screw-pitch error compensation is:

$$(+3)+(-7)+(+2)=(-2)$$

#### 5. Methods for compensation value setting

As shown in the above section, the compensation values are set according to the following factors.

1. Position relations between the reference point and compensation point
2. Machine moving direction
3. Compensation interval

The compensation value of the compensation point N ( $N=0, 1, 2, 3, \dots, 255$ ) is decided by the mechanical error (the differences correspond to the movement command value and movement) at the interval N, N-1.

#### ★ Methods for inputting compensation value

Using MDI&LCD or I/O interface, and the methods of inputting common parameters, compensation value can be input.

#### ★ Output of the compensation value

Using the same methods of outputting common parameters, the axes compensation value can be output.

#### ★ Compensation value backup

Methods for screw-pitch compensation value backup (including input and output) are identical with that of the parameter value backup.

#### ★ Cautions for compensation setting

- a) Compensation interval setting (Parameter 41, 42)
  - When compensation interval is positive, use this value for compensation.
  - When compensation interval is 0, the compensation is not performed.
- b) After setting the screw-pitch error parameter, return to the reference point.
- c) Screw-pitch error compensation value (compensation parameter 000~255)

# APPENDIX



## Appendix I Parameter List

### 1.1 Explanations for parameter

Parameters can be divided into the following ones according to the data types:

| Data type      | Active data range                    | Remarks                                                                               |
|----------------|--------------------------------------|---------------------------------------------------------------------------------------|
| Bit parameter  | 0 or 1                               | CNC provides default value. User can alter the setting as required.                   |
| Data parameter | Depend on the range of the parameter | CNC provides default range and default value, user can alter the setting as required. |
|                |                                      |                                                                                       |

1. For bit type and axis type parameters, each of figures consists of 8 bits. Each bit with different significance.
2. In the above table, the value of each kind of data is common active range. The specific parameter value range is not the same.

#### [Example]

(1) Significance of the bit parameter

|                          |                          |                          |                          |                          |                          |                          |                          |                          |
|--------------------------|--------------------------|--------------------------|--------------------------|--------------------------|--------------------------|--------------------------|--------------------------|--------------------------|
| <input type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> |
| Data No.                 | BIT7                     | BIT6                     | BIT5                     | BIT4                     | BIT3                     | BIT2                     | BIT1                     | BIT0                     |

(2) Significance of the data parameter

|                                |                                |                                |                      |
|--------------------------------|--------------------------------|--------------------------------|----------------------|
| <input type="text" value="0"/> | <input type="text" value="2"/> | <input type="text" value="1"/> | <input type="text"/> |
| Data No.                       | Data                           |                                |                      |

**Note:** The blank parameter numbers in the parameter explanation and the ones displayed on the screen but not written on the parameter list are prepared for future expansion. They should be set to 0.

### 1.2 Bit parameter

Parameter No.

|                                |                                |                                |     |     |  |      |      |     |  |  |
|--------------------------------|--------------------------------|--------------------------------|-----|-----|--|------|------|-----|--|--|
| <input type="text" value="0"/> | <input type="text" value="0"/> | <input type="text" value="1"/> | LAN | SCW |  | MDSP | WHLA | RAD |  |  |
|--------------------------------|--------------------------------|--------------------------------|-----|-----|--|------|------|-----|--|--|

- LAN** =1: English (display in English)  
 =0: Chinese (display in Chinese)
- SCW** =1: Inch system for min. command increment  
 =0: Metric system for min. command increment
- MDSP** =1: Spindle is analog volume control  
 =0: Other mode
- WHLA** =1: MPG mode  
 =0: Step mode
- RAD** =1: X-axis radius programming

=0: X axis diameter programming

Standard setting: 0 0 0 0 0 0 0 0

Parameter No.

|   |   |   |
|---|---|---|
| 0 | 0 | 2 |
|---|---|---|

|     |  |     |  |      |      |      |      |
|-----|--|-----|--|------|------|------|------|
| AUN |  | ORC |  | CCMP | SCRP | RDRN | PTSR |
|-----|--|-----|--|------|------|------|------|

**AUN** = 1: Add sequence number automatically in editing

= 0: Not add sequence number automatically in editing

**ORC** = 1: Tool offset by radius

= 0: Tool offset by diameter

**CCMP** = 1: Have tool compensation C function

= 0: Do not have compensation C function

**SCRP** = 1: Have screw-pitch compensation function

= 0: Do not have screw-pitch compensation function

**RDRN** = 1: Rapid traverse command is active in Dry run

= 0: Rapid traverse command is inactive in Dry run

**PTSR** = 1: Tool compensation by coordinate offset

= 0: Tool compensation by move

Standard setting: 0 0 0 0 0 0 1 1

Parameter No.

|   |   |   |
|---|---|---|
| 0 | 0 | 3 |
|---|---|---|

|  |     |      |      |      |      |     |     |
|--|-----|------|------|------|------|-----|-----|
|  | MNT | DECZ | DECX | DRDY | PCMD | HWC | SMT |
|--|-----|------|------|------|------|-----|-----|

**MNT** = 1: Machining time and parts number are not reserved when power off

= 0: Machining time and parts number are reserved when power off

**DECZ** = 1: Deceleration signal set to 1 indicates deceleration when Z axis returning to the reference point

= 0: Deceleration signal set to 0 indicates deceleration when Z axis returning to the reference point

**DECX** = 1: Deceleration signal set to 1 indicates deceleration when X axis returning to the reference point

= 0: Deceleration signal set to 0 indicates deceleration when X axis returning to the reference point

**DRDY** = 1: Drive unit/servo are ready when the signal is 1

= 0: Drive unit/servo are ready when the signal is 0

**PCMD** = 1: CNC output wave is pulse

= 0: CNC output wave is square

**HWC** = 1: MPG controls dry run interpolation

= 0: Common dry run

**SMT** = 1: lap transition precise and to the point

= 0: lap transition smooth

**Note::** When HWC parameter is active, after dry run is started, machine movement speed depends on the rotation speed of the MPG.

Standard setting: 0 0 0 0 0 1 0 0

Parameter No.

|   |   |   |
|---|---|---|
| 0 | 0 | 4 |
|---|---|---|

|     |     |     |     |       |      |     |     |
|-----|-----|-----|-----|-------|------|-----|-----|
| OFT | SMA | M30 | PPD | G50RP | CM98 | HDZ | HDX |
|-----|-----|-----|-----|-------|------|-----|-----|

- OFT** =1: offset power off do not memory  
 =0: offset power off memory
- SMA** =1: Spindle manual gearing for S code  
 =0: Spindle automatic gearing for S code
- M30** =1: Cursor to beginning when M30 executing  
 =0: Cursor to beginning when cycle start after M30 is executed
- PPD** =1: Relative coordinate is set by G50 in absolute zero programming  
 =0: Relative coordinate is not set by G50 in absolute zero programming
- G50RP** =1: G50 reference tool nose position, that is the current coordinate system is set directly when absolute coordinate system is set by G50  
 =0: G50 reference tool nose position, that is tool offset value is added as the current absolute coordinate when absolute coordinate system is set by G50

**CM98** =0: Input M, S, T codes other than standard ones of CNC, corresponding alarm will be issued in CNC

=1: Input M, S, T codes other than standard ones of CNC, CNC will not issue alarm, but automatically call a corresponding subprogram

1.1 : M code, When M codes other than standard ones of CNC are executed, the subprogram called is:

M○○: call subprogram 90○○.

1.2 : S code, when CNC performs S10~S99, the subprogram called are:

S□□ : call subprogram 91□□

**Note :** When spindle analog voltage is output, S code does not call the subprogram.

1.3 : Tcode, when CNC performs T10~T99, the subprogram called are:

T△△ : call subprogram 92△△.

**Note 1:** When M, S, T codes other than standard ones of CNC, corresponding subprogram should be input , otherwise, 078 alarm will be issued.

**Note 2:** Non-standard M, S, T codes can not be performed in MDI mode. Otherwise, M/S or T alarm will be issued.

**Note 3:**

**HDZ** =1: In manual mode, it is the positive direction of the Z-axis by moving left arrow key  
 =0: In manual mode, it is the positive direction of the Z-axis by moving right arrow key

**HDX** =1: In manual mode, it is the positive direction of the X-axis by moving up arrow key  
 =0: In manual mode, it is the positive direction of the X-axis by moving down arrow key

**Note:** If the direction of the manual arrow key on the panel is different from that of the actual moving direction of the machine, alter the direction of the selected axis of this parameter.

Standard setting: 0 0 1 1 0 0 0 0

Parameter No.

|   |   |   |      |      |       |     |     |  |     |     |
|---|---|---|------|------|-------|-----|-----|--|-----|-----|
| 0 | 0 | 5 | WSFT | ISOT | USEJK | CMZ | CPZ |  | ZMZ | ZMX |
|---|---|---|------|------|-------|-----|-----|--|-----|-----|

**WHFT** =1: Workpiece coordinate translation is active

=0: Workpiece coordinate translation is inactive

**ISOT** =1: After power-on or emergency stop, and reference point is not returned, manual rapid is active

=0: After power-on or emergency stop, and reference point is not returned, manual rapid is inactive

**USEJK** =1: use JK retract valid  
 =0: use JK retract invalid

**CMZ** =1: Machine zero return is not allowed  
 =0: Machine zero return is allowed

**CPZ** =1: Program zero return is not allowed  
 =0: Program zero return is allowed

**ZMZ** =1: When power on, direction of the Z-axis reference return and initial backlash direction is positive  
 =0: When power on, direction of the Z-axis reference return and initial backlash direction is negative

**ZMX** =1: When power on, direction of the X-axis reference return and initial backlash direction is positive  
 =0: When power on, direction of the X-axis reference return and initial backlash direction is negative

Standard setting: 0 1 0 0 0 0 1 1

Parameter No.

|   |   |   |  |      |     |     |  |  |      |      |
|---|---|---|--|------|-----|-----|--|--|------|------|
| 0 | 0 | 6 |  | APRS | ZCZ | ZCX |  |  | RTMZ | RTMX |
|---|---|---|--|------|-----|-----|--|--|------|------|

**APRS** =1: Coordinate system is set automatically after returning to the reference point  
 =0: Coordinate system is not set after returning to the reference point

**ZCZ** =1: Z-axis is magnetic switch zero return type C  
 =0: Deceleration switch and zero signal are necessary for machine zero return, return type B

**ZCX** =1: X-axis is magnetic switch zero return type C  
 =0: Deceleration switch and zero signal are necessary for machine zero return, return type B

**RMTZ** =1: Z-axis with machine zero point  
 =0: Z-axis without machine zero point

**RMTX** =1: X-axis with machine zero point  
 =0: X-axis without machine zero point

Standard setting: 0 1 0 0 0 0 0 0

Parameter No.

|   |   |   |      |       |  |      |      |  |      |      |
|---|---|---|------|-------|--|------|------|--|------|------|
| 0 | 0 | 7 | HAND | HDSTL |  | BDEC | RCUR |  | DIRZ | DIRX |
|---|---|---|------|-------|--|------|------|--|------|------|

**HAND** =1: MPG CW rotation is positive direction  
 =0: MPG CW rotation is negative direction

**HDSTL** =1: Handhold unit  
 =0: Common MPG

**BDEC** =1: backlash compensation at the speed of ascending and descending  
 =0: backlash compensation at the steady rate

**RCUR** =1: cursor return to beginning of program by pressing RESTE  
 =0: cursor do not return to beginning of program by pressing RESTE

**DIRZ** =1: Z-axis motor rotation direction is positive  
 =0: Z-axis motor rotation direction is negative

**DIRX** =1: X-axis motor rotation direction is positive  
 =0: X-axis motor rotation direction is negative

Standard setting: 0 0 0 0 0 0 0 0

Parameter No.

|   |   |   |
|---|---|---|
| 0 | 0 | 8 |
|---|---|---|

|      |     |      |      |      |      |      |      |
|------|-----|------|------|------|------|------|------|
| CLRT | CKR | LTVZ | LTVX | LMTZ | LMTX | ALMZ | ALMX |
|------|-----|------|------|------|------|------|------|

**CLRT** =1: for tool offset 00-064, press XYZ, invalid.

=0: for tool offset 00-064, press XYZ, valid.

**CKR** =1: Arc do not define R, alarm.

=0: Arc do not define R, no alarm

**LTVZ** =1: Z-axis hardware limit high level is active

=0: Z-axis hardware limit low level is active

**LTVX** =1: X-axis hardware limit high level is active

=0: X-axis hardware limit low level is active

**LMTZ** =1: Detect Z-axis hardware limit

=0: Not detect Z-axis hardware limit

**LMTX** =1: Detect X-axis hardware limit

=0: Not detect X-axis hardware limit

**ALMZ** =1: Z-axis drive unit alarm level is low level

=0: Z-axis drive unit alarm level is high level

**ALMX** =1: X-axis drive unit alarm level is low level

=0: X-axis drive unit alarm level is high level

Standard setting: 11 0 0 0 0 1 1

Parameter No.

|   |   |   |
|---|---|---|
| 0 | 0 | 9 |
|---|---|---|

|  |      |  |  |      |       |      |      |
|--|------|--|--|------|-------|------|------|
|  | ACC3 |  |  | NOFC | DOFSI | TSGN | TCPS |
|--|------|--|--|------|-------|------|------|

**ACC3** =1: rear linear acc/dec

**NOFC** =0: Input offset is inactive in counting

=1: Input offset is active in counting

**DOFSI** =0: Input offset is inactive in tool offset measure mode

=1: Input offset is active in tool offset measure mode

**TSGN** =1: Tool post in-position low level is active

=0: Tool post in-position high level is active

**TCPS** =1: Tool clamping signal high level is active

=0: Tool clamping signal low level is active

Standard setting: 0 0 0 0 1 1 1 0

Parameter No.

|   |   |   |
|---|---|---|
| 0 | 1 | 0 |
|---|---|---|

|      |      |  |      |     |     |      |     |
|------|------|--|------|-----|-----|------|-----|
| AGER | AGIN |  | AGIM | SPT | TRD | G0&T | TSS |
|------|------|--|------|-----|-----|------|-----|

**AGER** =0: Spindle automatic gearing is inactive

=1: Spindle automatic gearing is active

**AGIN** =0: Not detect in-position signal when shifting to gear 1,2

=1: Detect in-position signal when shifting to gear 1,2

**AGIM** =1: Gearing in-position signal is high level active

=0: Gearing in-position signal is low level active

**SPT** =1: Spindle jog is active in Auto mode

=0: Spindle jog is inactive in Auto mode

**TRD** =1: Detect tool post in-position

=0: Not detect tool post in-position  
**G0&T** =1: G0 and T codes can not be executed simultaneously  
 =0: G0 and T codes can be executed simultaneously  
**TSS** =1: Line-up tool post is selected  
 =0: Common tool post  
 Standard setting: 0 0 0 0 0 1 0 0

Parameter No.

|          |          |          |  |  |  |  |  |            |            |
|----------|----------|----------|--|--|--|--|--|------------|------------|
| <b>0</b> | <b>1</b> | <b>1</b> |  |  |  |  |  | <b>EAL</b> | <b>POD</b> |
|----------|----------|----------|--|--|--|--|--|------------|------------|

**EAL** =0: It is not programmable when alarming  
 =1: It is programmable when alarming  
**POD** =0: Program by decimal  
 =1: Not program by decimal  
 Standard setting: 0 0 0 0 0 0 0 1

Parameter No.

|          |          |          |             |             |  |  |            |            |  |             |
|----------|----------|----------|-------------|-------------|--|--|------------|------------|--|-------------|
| <b>0</b> | <b>1</b> | <b>2</b> | <b>WAR2</b> | <b>WAR1</b> |  |  | <b>WA2</b> | <b>WA1</b> |  | <b>RSJG</b> |
|----------|----------|----------|-------------|-------------|--|--|------------|------------|--|-------------|

**WAR2** =1: External alarm 2 is active  
 =0: External alarm 2 is inactive  
**WAR1** =1: External alarm 1 is active  
 =0: External alarm 1 is inactive  
**WA2** =1: External alarm 2 high level is active  
 =0: External alarm 2 low level is active  
**WA1** =1: External alarm 1 high level is active  
 =0: External alarm 1 low level is active  
**RSJG** =1: Outputting M03, M04, lubrication, cooling are not affected by pressing reset key.  
 =0: M03, M04, lubrication, cooling are stopped when pressing reset key  
 Standard setting: 0 0 0 0 0 0 0 0

Parameter No.

|          |          |          |              |             |            |            |             |            |             |             |
|----------|----------|----------|--------------|-------------|------------|------------|-------------|------------|-------------|-------------|
| <b>0</b> | <b>1</b> | <b>3</b> | <b>PLCKV</b> | <b>PLCK</b> | <b>M23</b> | <b>M21</b> | <b>CEIO</b> | <b>JLB</b> | <b>RTMC</b> | <b>SBKM</b> |
|----------|----------|----------|--------------|-------------|------------|------------|-------------|------------|-------------|-------------|

**PLCKV** =1: Program lock signal is high level active  
 =0: Program lock signal is low level active  
**PLCK** =1: Program lock signal is active  
 =0: Program lock signal is inactive  
**M23** =1: M23 is active  
 =0: M23 is inactive  
**M21** =1: M21 is active  
 =0: M21 is inactive  
**CEIO** =1: Expansion input IO port is closed after reset  
 =0: Expansion input IO port is not closed after reset  
**JLB** =1: Intermittent lubrication  
 =0: Continuous lubrication  
**RTMC** =1: Macro common variable is cleared after reset  
 =0: Macro common variable is not changed after reset  
**SBKM** =1: Single block is active when macro code is executing

=0: Single block is inactive when macro code is executing

Standard setting: 0 0 0 0 0 0 0 1

Parameter No.

|   |   |   |      |      |     |  |      |     |      |     |
|---|---|---|------|------|-----|--|------|-----|------|-----|
| 0 | 1 | 4 | KEY1 | SKEY | CHK |  | MESP | MOT | M@SP | MST |
|---|---|---|------|------|-----|--|------|-----|------|-----|

**KEY1** =1: Program switch is on when power on

=0: Program switch is off when power on

**SKEY** =1: Program switch is shielded

=0: Program switch is not shielded

**CHK** =1: Not check parameter tool offset

=0: Check parameter tool offset

**MESP** =1: Emergency stop signal (ESP) is shielded

=0: Emergency stop signal (ESP) is not shielded

**MOT** =1: Not check software stroke limit switch

=0: Check software stroke limit switch

**M@SP** =1: External signal (SP) is shielded

=0: External signal (SP) is not shielded

**MST** =1: External signal (ST) is shielded

=0: External signal (ST) is not shielded

Standard setting: 1 0 0 0 1 1 1 1

Parameter No.

|   |   |   |      |      |      |      |     |      |      |       |
|---|---|---|------|------|------|------|-----|------|------|-------|
| 0 | 1 | 5 | STAR | OWAR | OM30 | OMST | WZ2 | KPDW | SPEN | SPENC |
|---|---|---|------|------|------|------|-----|------|------|-------|

**STAR** =1: Spindle star delta starting is active

=0: Spindle star delta starting is inactive

**OWAR** =1: Output alarm signal is active (indicator shall be used to display the CNC is in alarm state)

=0: Output alarm signal is inactive

**OM30** =1: Output M30 signal is active

=0: Output M30 signal is inactive

**OMST** =1: Output STM signal is active

=0: Output STM signal is inactive

**WZ2** =1: Tail stock hierarchical advancing/retracing control is active

=0: Tail stock hierarchical advancing/retracing control is inactive

**KPDW** =1: Check chuck in-position signal

=0: Not check chuck in-position signal

**SPEN** =1: Feed and spindle enable input are active

=0: Feed and spindle enable input are inactive

**SPENC** =1: Control feed and spindle

=0: Control spindle

**Note:** When SPENC and SPEN are 0, spindle enable function is inactive. When SPENC is 0, SPEN is 1, spindle rotation enable function is inactive. When SPENC and SPEN are 1, spindle and feed are enabled if SAR signal is connected. While SAR signal is not connected, spindle is enabled, feed is forbidden when SPEN signal is connected. When SAR and SPEN signals are not connected, spindle and feed are forbidden.

Standard setting: 0 0 0 0 0 0 0 0

Parameter No.

|   |   |   |      |      |      |      |      |      |      |      |
|---|---|---|------|------|------|------|------|------|------|------|
| 0 | 1 | 6 | NWKP | SFDV | SFDR | PSCK | PSCV | SLTW | SLSP | SLQP |
|---|---|---|------|------|------|------|------|------|------|------|

**NWKP** =1: Outer chuck mode of the chuck

=0: Inner chuck mode of the chuck

**SFDV** =1: Protection door is high level active

=0: Protection door is low level active

**SFDR** =1: Check protection door in cycle start, operation is not allowed when protection door is open

=0: Not check protection door in cycle start

**PSCK** =1: Check pressure signal in cycle start, it is not allowed

=0: Not check pressure signal in cycle start

**PSCV** =1: Pressure detect signal is high level active

=0: Pressure detect signal is low level active

**SLTW** =1: Tail stock function is active

=0: Tail stock function is inactive

**SLSP** =1: Detect chuck clamping when spindle starts

=0: Not detect chuck state when spindle starts

**SLQP** =1: Chuck function is active

=0: Chuck function is inactive

Standard setting: 0 0 0 0 0 0 0 0

### 1.3. Data Parameter

#### GSK980TB3/ TB3-V

| Parameter No. | Parameter definition                            | Default value |
|---------------|-------------------------------------------------|---------------|
| 0017          | X-axis coordinate command multiplier ratio CMRX | 1             |

|      |                                                 |   |
|------|-------------------------------------------------|---|
| 0018 | Z-axis coordinate command multiplier ratio CMRZ | 1 |
|------|-------------------------------------------------|---|

CMRX, CMRZ: Command multiplier ratio of each coordinate, which is related to gear ratio, screw-pitch error and pulse equivalent.

Setting range: 1~65535

|      |                                                    |   |
|------|----------------------------------------------------|---|
| 0019 | X-axis command frequency division coefficient CMDX | 1 |
|------|----------------------------------------------------|---|

|      |                                                    |   |
|------|----------------------------------------------------|---|
| 0020 | Z-axis command frequency division coefficient CMDZ | 1 |
|------|----------------------------------------------------|---|

CMDX, CMDZ: Frequency division coefficient of each axis, which is related to gear ratio, screw-pitch error and pulse equivalent.

Setting range: 1~65535

|      |                                     |                         |
|------|-------------------------------------|-------------------------|
| 0021 | X-axis rapid traverse rate (mm/min) | Step 3000<br>Servo 4000 |
|------|-------------------------------------|-------------------------|

Setting range: 30~30000 (Maximum value shall not exceed 15000 in diameter programming. In diameter programming, the actual speed is twice of the specified value)

|      |                                     |                        |
|------|-------------------------------------|------------------------|
| 0022 | Z-axis rapid traverse rate (mm/min) | Step6000<br>Servo 8000 |
|------|-------------------------------------|------------------------|

Setting range: 30~30000

|      |                                                               |                     |
|------|---------------------------------------------------------------|---------------------|
| 0023 | X-axis linear ACC&DEC time constant (used for rapid traverse) | Step200<br>Servo 60 |
|------|---------------------------------------------------------------|---------------------|

Setting range: 8~4000 (Set above 300 when using stepping drive unit)

|      |                                                               |                     |
|------|---------------------------------------------------------------|---------------------|
| 0024 | Z-axis linear ACC&DEC time constant (used for rapid traverse) | Step200<br>Servo 60 |
|------|---------------------------------------------------------------|---------------------|

Setting range: 8~4000 (Set above 300 when using stepping drive unit)

|      |                                                          |    |
|------|----------------------------------------------------------|----|
| 0025 | X-axis S ACC&DEC time constant (used for rapid traverse) | 60 |
|------|----------------------------------------------------------|----|

Setting range: 8~4000 (Set above 300 when using stepping drive unit)

|      |                                                          |    |
|------|----------------------------------------------------------|----|
| 0026 | Z-axis S ACC&DEC time constant (used for rapid traverse) | 60 |
|------|----------------------------------------------------------|----|

Setting range: 8~4000 (Set above 300 when using stepping drive unit)

|      |                                         |      |
|------|-----------------------------------------|------|
| 0027 | Upper speed of cutting feeding (mm/min) | 8000 |
|------|-----------------------------------------|------|

Upper speed of cutting feeding applies to all coordinates.

Setting range: 0~8000 Unit: mm/min

|      |                                                               |     |
|------|---------------------------------------------------------------|-----|
| 0028 | Lower value of the ACC&DEC low speed in cutting feed (mm/min) | 200 |
|------|---------------------------------------------------------------|-----|

Setting range: 0~8000 Unit: mm/min

|      |                                              |                     |
|------|----------------------------------------------|---------------------|
| 0029 | S acceleration time constant in cutting feed | Step200<br>Servo 60 |
|------|----------------------------------------------|---------------------|

Setting range: 8~4000 (Set above 300 when using stepping drive unit)

|      |                                              |                     |
|------|----------------------------------------------|---------------------|
| 0030 | S deceleration time constant in cutting feed | Step200<br>Servo 60 |
|------|----------------------------------------------|---------------------|

Setting range: 8~4000 (Set above 300 when using stepping drive unit)

|      |                                              |                     |
|------|----------------------------------------------|---------------------|
| 0031 | Linear ACC&DEC time constant in cutting feed | Step200<br>Servo 60 |
|------|----------------------------------------------|---------------------|

Setting range: 8~4000 (Set above 300 when using stepping drive unit)

|      |                                             |     |
|------|---------------------------------------------|-----|
| 0032 | Minimum speed Fo of rapid override (mm/min) | 100 |
|------|---------------------------------------------|-----|

Setting range: 0~8000 Unit: mm/min

|      |                                                     |    |
|------|-----------------------------------------------------|----|
| 0033 | Pulse frequency of backlash offset (Shared by axes) | 80 |
|------|-----------------------------------------------------|----|

Setting range: 1~16384 Unit: Hz (Set to 5 when servo motor is fixed. Set to 3 when stepping motor is fixed)

|      |                                       |    |
|------|---------------------------------------|----|
| 0034 | Speed of the backlash offset (mm/min) | 40 |
|------|---------------------------------------|----|

Setting range: 1~1000 Unit: mm/min

|      |                                            |    |
|------|--------------------------------------------|----|
| 0035 | Linear ACC&DEC time of the backlash offset | 40 |
|------|--------------------------------------------|----|

Setting range: 1~1000

|      |                          |   |
|------|--------------------------|---|
| 0036 | X-axis offset value (μm) | 0 |
|------|--------------------------|---|

Setting range: 0~65535

|      |                          |   |
|------|--------------------------|---|
| 0037 | Z-axis offset value (μm) | 0 |
|------|--------------------------|---|

Setting range: 0~65535

|      |                                                          |   |
|------|----------------------------------------------------------|---|
| 0038 | Screw-pitch error compensation override (Shared by axes) | 1 |
|------|----------------------------------------------------------|---|

Setting range: 1~8

|      |                                              |   |
|------|----------------------------------------------|---|
| 0039 | X-axis screw-pitch error compensation origin | 0 |
|------|----------------------------------------------|---|

Setting range: 0~255

|      |                                              |   |
|------|----------------------------------------------|---|
| 0040 | Z-axis screw-pitch error compensation origin | 0 |
|------|----------------------------------------------|---|

Setting range: 0~255

|      |                                               |      |
|------|-----------------------------------------------|------|
| 0041 | X-axis screw-pitch compensation interval (μm) | 1000 |
|------|-----------------------------------------------|------|

Setting range: 100~9999999 Unit: μm

|      |                                               |      |
|------|-----------------------------------------------|------|
| 0042 | Z-axis screw-pitch compensation interval (μm) | 1000 |
|------|-----------------------------------------------|------|

Setting range: 100~9999999 Unit: μm

|      |                                                                      |     |
|------|----------------------------------------------------------------------|-----|
| 0043 | Low speed when returning to the reference point, (FL) speed (mm/min) | 200 |
|------|----------------------------------------------------------------------|-----|

Setting range: 6~600 Unit: mm/min

|      |                                                                            |   |
|------|----------------------------------------------------------------------------|---|
| 0044 | After reset, X value of coordinate system is set by CNC automatically (μm) | 0 |
|------|----------------------------------------------------------------------------|---|

After reset, CNC automatically sets the absolute coordinate value to the one specified by coordinate system

Setting range: ±9999999 Unit: μm

|      |                                                                            |   |
|------|----------------------------------------------------------------------------|---|
| 0045 | After reset, Z value of coordinate system is set by CNC automatically (μm) | 0 |
|------|----------------------------------------------------------------------------|---|

Setting range: ±9999999 Unit: μm

|      |                                   |         |
|------|-----------------------------------|---------|
| 0046 | X-axis positive stroke limit (μm) | 9999999 |
|------|-----------------------------------|---------|

Setting range: ±9999999 Unit: μm

|      |                                   |          |
|------|-----------------------------------|----------|
| 0047 | X-axis negative stroke limit (μm) | -9999999 |
|------|-----------------------------------|----------|

Setting range: ±9999999 Unit: μm

|      |                                   |         |
|------|-----------------------------------|---------|
| 0048 | Z-axis positive stroke limit (μm) | 9999999 |
|------|-----------------------------------|---------|

Setting range: ±9999999 Unit: μm

|      |                                   |          |
|------|-----------------------------------|----------|
| 0049 | Z-axis negative stroke limit (μm) | -9999999 |
|------|-----------------------------------|----------|

Setting range: ±9999999 Unit: μm

|      |                                                                               |    |
|------|-------------------------------------------------------------------------------|----|
| 0050 | In programming, increment for inserting program sequence number automatically | 10 |
|------|-------------------------------------------------------------------------------|----|

Setting range: 0~9999

|      |                              |     |
|------|------------------------------|-----|
| 0051 | Spindle analog adjusted data | 999 |
|------|------------------------------|-----|

Setting range: -999~999

|      |                                |      |
|------|--------------------------------|------|
| 0052 | Line number of spindle encoder | 1024 |
|------|--------------------------------|------|

Setting range: 1024. 1200

|      |                                                        |   |
|------|--------------------------------------------------------|---|
| 0053 | Gear ratio of spindle and encoder: spindle gear number | 1 |
|------|--------------------------------------------------------|---|

Setting range: 1~65535

|      |                                                        |   |
|------|--------------------------------------------------------|---|
| 0054 | Gear ratio of spindle and encoder: spindle gear number | 1 |
|------|--------------------------------------------------------|---|

Setting range: 1~65535

|      |                                                    |      |
|------|----------------------------------------------------|------|
| 0055 | When spindle speed is 10V, spindle speed at gear 1 | 9999 |
|------|----------------------------------------------------|------|

Setting range: 1~9999

|      |                                                    |      |
|------|----------------------------------------------------|------|
| 0056 | When spindle speed is 10V, spindle speed at gear 2 | 9999 |
|------|----------------------------------------------------|------|

Setting range: 1~9999

|      |                                                    |      |
|------|----------------------------------------------------|------|
| 0057 | When spindle speed is 10V, spindle speed at gear 3 | 9999 |
|------|----------------------------------------------------|------|

Setting range: 1~9999

|      |                                                    |      |
|------|----------------------------------------------------|------|
| 0058 | When spindle speed is 10V, spindle speed at gear 4 | 9999 |
|------|----------------------------------------------------|------|

Setting range: 1~9999

|      |                                                    |    |
|------|----------------------------------------------------|----|
| 0059 | Threading is performed after percentage is stable. | 95 |
|------|----------------------------------------------------|----|

Setting range: 0~100

|      |                                    |      |
|------|------------------------------------|------|
| 0060 | Spindle Max. programminG Codespeed | 9999 |
|------|------------------------------------|------|

Setting range: 1~9999

|      |                                                   |   |
|------|---------------------------------------------------|---|
| 0061 | Spindle analog adjusted data (Adjust it slightly) | 0 |
|------|---------------------------------------------------|---|

Setting range: -999~999. Suppose the spindle is accurate at high speed, if it is not accurate at low speed, adjust it to an appropriate value.

|      |                                      |                   |
|------|--------------------------------------|-------------------|
| 0062 | Spindle speed sampling period (x4ms) | Step20<br>Servo 1 |
|------|--------------------------------------|-------------------|

Setting range: 1~200

|      |                            |    |
|------|----------------------------|----|
| 0063 | Gear shifting time 1(x4ms) | 50 |
|------|----------------------------|----|

Setting range: 0~9999999 Unit: ×4ms

|      |                            |    |
|------|----------------------------|----|
| 0064 | Gear shifting time 2(x4ms) | 50 |
|------|----------------------------|----|

Setting range: 0~9999999 Unit: ×4ms

|      |                                                 |     |
|------|-------------------------------------------------|-----|
| 0065 | Max. cutting feed speed of each rotation (mm/r) | 500 |
|------|-------------------------------------------------|-----|

Setting range: 0~500 Unit: mm/r

|      |                                             |   |
|------|---------------------------------------------|---|
| 0066 | Rotational speed when spindle gear shifting | 0 |
|------|---------------------------------------------|---|

Setting range: 0~9999 Unit: r/min

|      |                                                                      |     |
|------|----------------------------------------------------------------------|-----|
| 0067 | Spindle lower speed under the constant surface speed control (r/min) | 100 |
|------|----------------------------------------------------------------------|-----|

Setting range: 0~9999 Unit: r/min

|      |                                           |    |
|------|-------------------------------------------|----|
| 0068 | Chamfering width of screw cutting (THDCH) | 10 |
|------|-------------------------------------------|----|

Setting range: 0~99 Chamfering width of screw=THDCH×1/10×screw-pitch

|      |                                                        |                     |
|------|--------------------------------------------------------|---------------------|
| 0069 | X-axis linear ACC&DEC constant in thread cutting (G92) | Step200<br>Servo 40 |
|------|--------------------------------------------------------|---------------------|

Setting range: 6~4000

|      |                                                                    |                     |
|------|--------------------------------------------------------------------|---------------------|
| 0070 | Lower value of S type ACC&DEC of each axis in thread cutting (G92) | Step200<br>Servo 40 |
|------|--------------------------------------------------------------------|---------------------|

Setting range: 6~4000 Unit: mm/min

|      |                                                  |   |
|------|--------------------------------------------------|---|
| 0071 | Cutting depth in multiple fixed cycle (G71, G72) | 0 |
|------|--------------------------------------------------|---|

Setting range: 0~9999999 Unit: 0.001mm

|      |                                                    |   |
|------|----------------------------------------------------|---|
| 0072 | Tool retraction in multiple fixed cycle (G71, G72) | 0 |
|------|----------------------------------------------------|---|

Setting range: 0~9999999 Unit: 0.001mm

|      |                                                           |   |
|------|-----------------------------------------------------------|---|
| 0073 | X-axial total cutting value in multiple fixed cycle (G73) | 0 |
|------|-----------------------------------------------------------|---|

Setting range: -9,999,999~9,999,999 Unit: 0.001mm

|      |                                                           |   |
|------|-----------------------------------------------------------|---|
| 0074 | Z-axial total cutting value in multiple fixed cycle (G73) | 0 |
|------|-----------------------------------------------------------|---|

Setting range: -9,999,999~9,999,999 Unit: 0.001mm

|      |                                                   |   |
|------|---------------------------------------------------|---|
| 0075 | Cycle cutting times in multiple fixed cycle (G73) | 0 |
|------|---------------------------------------------------|---|

Setting range: 0~9999999 Unit: times

|      |                                                    |   |
|------|----------------------------------------------------|---|
| 0076 | Tool retraction in multiple fixed cycle (G74, G75) | 0 |
|------|----------------------------------------------------|---|

Setting range: 0~9999999 Unit: 0.001mm

|      |                                                              |   |
|------|--------------------------------------------------------------|---|
| 0077 | Repetitions of finish matching in multiple fixed cycle (G76) | 0 |
|------|--------------------------------------------------------------|---|

Setting range: 0~9999999 Unit: time

|      |                                               |   |
|------|-----------------------------------------------|---|
| 0078 | Tool nose angle in multiple fixed cycle (G76) | 0 |
|------|-----------------------------------------------|---|

Setting range: 0, 29, 30, 55, 60, 80

|      |                                                  |   |
|------|--------------------------------------------------|---|
| 0079 | Min. cutting depth in multiple fixed cycle (G76) | 0 |
|------|--------------------------------------------------|---|

Setting range: 0~9999999 Unit: 0.001mm

|      |                                                |   |
|------|------------------------------------------------|---|
| 0080 | Finish allowance in multiple fixed cycle (G76) | 0 |
|------|------------------------------------------------|---|

Setting range: 0~9999999 Unit: 0.001mm

|      |                             |   |
|------|-----------------------------|---|
| 0081 | Total tool number selection | 4 |
|------|-----------------------------|---|

Setting range: 0~8

|      |                              |    |
|------|------------------------------|----|
| 0082 | Tool changing time T1 (x4ms) | 30 |
|------|------------------------------|----|

Setting range: 0~9999999 Unit: ×4ms

|      |                                                      |      |
|------|------------------------------------------------------|------|
| 0083 | Upper time for moving a tool in tool changing (x4ms) | 1000 |
|------|------------------------------------------------------|------|

Setting range: 0~9999999 Unit: ×4ms

|      |                                                                        |       |
|------|------------------------------------------------------------------------|-------|
| 0084 | Upper time from changing the 1st tool to changing the last tool (x4ms) | 15000 |
|------|------------------------------------------------------------------------|-------|

Setting range: 0~9999999 Unit: ×4ms

|      |                                            |     |
|------|--------------------------------------------|-----|
| 0085 | Tool post CW rotation clamping time (x4ms) | 250 |
|------|--------------------------------------------|-----|

Setting range: 0~9999999 Unit: ×4ms

|      |                              |   |
|------|------------------------------|---|
| 0086 | M code execution time (x4ms) | 1 |
|------|------------------------------|---|

Setting range: 0~9999999 Unit: ×4ms

|      |                              |   |
|------|------------------------------|---|
| 0087 | S code execution time (x4ms) | 1 |
|------|------------------------------|---|

Setting range: 0~9999999 Unit: ×4ms

|      |                                                |     |
|------|------------------------------------------------|-----|
| 0088 | Alarm time of not receiving *TCP signal (x4ms) | 500 |
|------|------------------------------------------------|-----|

Setting range: 0~9999999 Unit: ×4ms

|      |                                                                   |     |
|------|-------------------------------------------------------------------|-----|
| 0089 | Duration from spindle command stop to spindle brake output (x4ms) | 100 |
|------|-------------------------------------------------------------------|-----|

Setting range: 0~9999999 Unit: ×4ms

|      |                                  |     |
|------|----------------------------------|-----|
| 0090 | Spindle brake output time (x4ms) | 100 |
|------|----------------------------------|-----|

Setting range: 0~9999999 Unit: ×4ms

|      |                                 |     |
|------|---------------------------------|-----|
| 0091 | Chuck clamping in-position time | 100 |
|------|---------------------------------|-----|

Setting range: 0~9999999 Unit: ×4ms

|      |                                           |     |
|------|-------------------------------------------|-----|
| 0092 | Setting of pressure detection time (x4ms) | 100 |
|------|-------------------------------------------|-----|

Setting range: 0~9999999 Unit: ×4ms

|      |                                             |     |
|------|---------------------------------------------|-----|
| 0093 | Star triangle starting time constant (x4ms) | 100 |
|------|---------------------------------------------|-----|

Setting range: 0~9999999 Unit: ×4ms

|      |                               |     |
|------|-------------------------------|-----|
| 0094 | Spindle jog dwell time (x4ms) | 100 |
|------|-------------------------------|-----|

Setting range: 0~9999999 Unit: ×4ms

|      |                                   |      |
|------|-----------------------------------|------|
| 0095 | Baud rate of serial communication | 9600 |
|------|-----------------------------------|------|

Setting range: 4800. 9600. 19200. 38400. 57600. 115200

|      |                                 |                      |
|------|---------------------------------|----------------------|
| 0096 | (x4ms)Page starting time (x4ms) | Step600<br>Servo 100 |
|------|---------------------------------|----------------------|

Setting range: 0~9999 Unit: ×4ms

|      |                                             |   |
|------|---------------------------------------------|---|
| 0097 | Duration time for signal removing vibration | 2 |
|------|---------------------------------------------|---|

Setting range: 2~15 (Do not change standard setting without special circumstances)

|      |                                      |      |
|------|--------------------------------------|------|
| 0098 | Error limit of circular arc 2*R (um) | 2000 |
|------|--------------------------------------|------|

Setting range: 1~9999999

|      |           |   |
|------|-----------|---|
| 0099 | Undefined | 0 |
|------|-----------|---|

|      |           |   |
|------|-----------|---|
| 0100 | Undefined | 0 |
|------|-----------|---|

|      |                                                                                              |                     |
|------|----------------------------------------------------------------------------------------------|---------------------|
| 0101 | Tool retraction error adjustment value of G33 tapping<br>(adjustable range from -400 to 400) | Step-45<br>Servo 12 |
|------|----------------------------------------------------------------------------------------------|---------------------|

Tool retraction error adjustment parameter G33 of tapping, which is adjusted and used according to the actual circumstances (converter spindle, gear spindle, matching with stepping drive or servo drive etc. are fixed). In principle, turn it up if tool retraction lags behind, and turn it down if tool retraction performs ahead of time.

Setting range: -400~400, no unit

|      |                                                     |    |
|------|-----------------------------------------------------|----|
| 0102 | Degree of corner between lines in post ACC&DEC mode | 30 |
|------|-----------------------------------------------------|----|

The smaller the precision is, the higher the efficiency is, and vice versa. The angle is the biggest when it is 0%, while there is no angle when it is 100%. Namely, it is a linear. Corners between lines are formed automatically in post ACC&DEC mode, which are affected by ACC&DEC time and current speed. When non-mark connection is required, set this parameter to 0%.

Setting range: 0~100

|      |                                   |    |
|------|-----------------------------------|----|
| 0103 | Initial speed of cutting (mm/min) | 40 |
|------|-----------------------------------|----|

After power on, interpolation speed is not specified by the program when interpolation command is being executed, perform interpolation at speed specified by this parameter. When this parameter is 0, and the speed is not specified by the program, alarm will be issued by CNC.

Setting range: 0~2000

|      |                                                           |    |
|------|-----------------------------------------------------------|----|
| 0104 | Reversing speed of reference return in C mode<br>(mm/min) | 45 |
|------|-----------------------------------------------------------|----|

In return type C, when returning to reference point, it touches deceleration block and decelerates. After departing from the deceleration block, it continuously travels short distance and then returns. The parameter sets the reversing speed. The bigger the value is, the worse the precision is.

Setting range: 0~1000 Unit: mm/min

|      |                                                  |   |
|------|--------------------------------------------------|---|
| 0105 | M12, M13 routine output/pulse output time (x4ms) | 0 |
|------|--------------------------------------------------|---|

When the chuck control signal is controlled by pulse, set signal output time by this parameter. If the specified time is delayed, CNC cancels outputting signal, while the state of the CNC is not changed. When this parameter is set to 0, the signal is long output signal until the command is changed.

Setting range: 0~9999999 Unit: x4ms

|      |           |   |
|------|-----------|---|
| 0106 | Undefined | 0 |
|------|-----------|---|

|      |                               |      |
|------|-------------------------------|------|
| 0107 | Intermittent lubrication time | 1000 |
|------|-------------------------------|------|

Setting range: 0~9999999 Unit: ×4ms

|      |                                        |      |
|------|----------------------------------------|------|
| 0108 | Pause time of intermittent lubrication | 1000 |
|------|----------------------------------------|------|

Setting range: 0~9999999 Unit: ×4ms

|      |                                                               |      |
|------|---------------------------------------------------------------|------|
| 0109 | Rapid speed when returning to the reference point<br>(mm/min) | 6000 |
|------|---------------------------------------------------------------|------|

Setting range: 0~9999999

|      |           |   |
|------|-----------|---|
| 0110 | Undefined | 0 |
| 0111 | Undefined | 0 |

|      |           |   |
|------|-----------|---|
| 0112 | Undefined | 0 |
|------|-----------|---|

By setting the following parameter, it is able to set whether the output signal is pulse or level. 0: level Non zero: pulse

|      |                        |   |
|------|------------------------|---|
| 0113 | S01 output time (x4ms) | 0 |
|------|------------------------|---|

Setting range: 0~9999999 Unit: ×4ms

|      |                        |   |
|------|------------------------|---|
| 0114 | S02 output time (x4ms) | 0 |
|------|------------------------|---|

Setting range: 0~9999999 Unit: ×4ms

|      |                        |   |
|------|------------------------|---|
| 0115 | S03 output time (x4ms) | 0 |
|------|------------------------|---|

Setting range: 0~9999999 Unit: ×4ms

|      |                        |   |
|------|------------------------|---|
| 0116 | S04 output time (x4ms) | 0 |
|------|------------------------|---|

Setting range: 0~9999999 Unit: ×4ms

|      |                        |   |
|------|------------------------|---|
| 0117 | M03 output time (x4ms) | 0 |
|------|------------------------|---|

Setting range: 0~9999999 Unit: ×4ms

|      |                        |   |
|------|------------------------|---|
| 0118 | M04 output time (x4ms) | 0 |
|------|------------------------|---|

Setting range: 0~9999999 Unit: ×4ms

|      |                        |   |
|------|------------------------|---|
| 0119 | M05 output time (x4ms) | 0 |
|------|------------------------|---|

Setting range: 0~9999999 Unit: ×4ms

|      |             |   |
|------|-------------|---|
| 0120 | To be added | 0 |
|------|-------------|---|

Setting range: 0~9999999 Unit: ×4ms

|      |                        |   |
|------|------------------------|---|
| 0121 | M08 output time (x4ms) | 0 |
|------|------------------------|---|

Setting range: 0~9999999 Unit: ×4ms

|      |                        |   |
|------|------------------------|---|
| 0122 | M10 output time (x4ms) | 0 |
|------|------------------------|---|

Setting range: 0~9999999 Unit: ×4ms

|      |                        |   |
|------|------------------------|---|
| 0123 | M11 output time (x4ms) | 0 |
|------|------------------------|---|

Setting range: 0~9999999 Unit: ×4ms

|      |             |   |
|------|-------------|---|
| 0124 | To be added | 0 |
|------|-------------|---|

Setting range: 0~9999999 Unit: ×4ms

|      |             |   |
|------|-------------|---|
| 0125 | To be added | 0 |
|------|-------------|---|

Setting range: 0~9999999 Unit: ×4ms

|      |                        |   |
|------|------------------------|---|
| 0126 | M30 output time (x4ms) | 0 |
|------|------------------------|---|

Setting range: 0~9999999 Unit: ×4ms

|      |                        |   |
|------|------------------------|---|
| 0127 | M32 output time (x4ms) | 0 |
|------|------------------------|---|

Setting range: 0~9999999 Unit: ×4ms

|      |             |   |
|------|-------------|---|
| 0128 | To be added | 0 |
|------|-------------|---|

Setting range: 0~9999999 Unit: ×4ms

|      |             |   |
|------|-------------|---|
| 0129 | To be added | 0 |
|------|-------------|---|

Setting range: 0~9999999 Unit: ×4ms

|      |                        |   |
|------|------------------------|---|
| 0130 | STM output time (x4ms) | 0 |
|------|------------------------|---|

Setting range: 0~9999999 Unit: ×4ms

|      |                         |   |
|------|-------------------------|---|
| 0131 | WARN output time (x4ms) | 0 |
|------|-------------------------|---|

Setting range: 0~9999999 Unit: ×4ms

|      |                         |   |
|------|-------------------------|---|
| 0132 | STAR output time (x4ms) | 0 |
|------|-------------------------|---|

Setting range: 0~9999999 Unit: ×4ms

|      |                          |   |
|------|--------------------------|---|
| 0133 | TRIAN output time (x4ms) | 0 |
|------|--------------------------|---|

Setting range: 0~9999999 Unit: ×4ms

|      |                          |   |
|------|--------------------------|---|
| 0134 | TLOCK output time (x4ms) | 0 |
|------|--------------------------|---|

Setting range: 0~9999999 Unit: ×4ms

|      |                        |   |
|------|------------------------|---|
| 0135 | M51 output time (x4ms) | 0 |
|------|------------------------|---|

Setting range: 0~9999999 Unit: ×4ms

|      |                        |   |
|------|------------------------|---|
| 0136 | M53 output time (x4ms) | 0 |
|------|------------------------|---|

Setting range: 0~9999999 Unit: ×4ms

|      |                        |   |
|------|------------------------|---|
| 0137 | M55 output time (x4ms) | 0 |
|------|------------------------|---|

Setting range: 0~9999999 Unit: ×4ms

|      |                        |   |
|------|------------------------|---|
| 0138 | M57 output time (x4ms) | 0 |
|------|------------------------|---|

Setting range: 0~9999999 Unit: ×4ms

|      |                        |   |
|------|------------------------|---|
| 0139 | M59 output time (x4ms) | 0 |
|------|------------------------|---|

Setting range: 0~9999999 Unit: ×4ms

|      |                        |   |
|------|------------------------|---|
| 0140 | M61 output time (x4ms) | 0 |
|------|------------------------|---|

Setting range: 0~9999999 Unit: ×4ms

|      |                        |   |
|------|------------------------|---|
| 0141 | M63 output time (x4ms) | 0 |
|------|------------------------|---|

Setting range: 0~9999999 Unit: ×4ms

|      |                        |   |
|------|------------------------|---|
| 0142 | M65 output time (x4ms) | 0 |
|------|------------------------|---|

Setting range: 0~9999999 Unit: ×4ms

|      |                        |   |
|------|------------------------|---|
| 0143 | M67 output time (x4ms) | 0 |
|------|------------------------|---|

Setting range: 0~9999999 Unit: ×4ms

|      |                        |   |
|------|------------------------|---|
| 0144 | M69 output time (x4ms) | 0 |
|------|------------------------|---|

Setting range: 0~9999999 Unit: ×4ms

|      |                                                                           |   |
|------|---------------------------------------------------------------------------|---|
| 0145 | G31 type: 0 is for standard; >0 is for expansion (see manual for details) | 0 |
|------|---------------------------------------------------------------------------|---|

Setting range: 0~9999999 Unit: ×4ms

|      |                                                                          |     |
|------|--------------------------------------------------------------------------|-----|
| 0146 | Number of G31 command signal input port and high/low level (default 151) | 151 |
|------|--------------------------------------------------------------------------|-----|

Setting range: 0~9999999 Unit: ×4ms

|      |             |   |
|------|-------------|---|
| 0147 | To be added | 0 |
|------|-------------|---|

Setting range: 0~9999999 Unit: ×4ms

|      |                          |   |
|------|--------------------------|---|
| 0148 | Tool post expansion type | 0 |
|------|--------------------------|---|

Setting range: 0~9999999

|      |             |   |
|------|-------------|---|
| 0149 | To be added | 0 |
|------|-------------|---|

Setting range: 0~9999999 Unit: ×4ms

|      |             |   |
|------|-------------|---|
| 0150 | To be added | 0 |
|------|-------------|---|

Setting range: 0~9999999 Unit: ×4ms

|      |             |   |
|------|-------------|---|
| 0151 | To be added | 0 |
|------|-------------|---|

Setting range: 0~9999999 Unit: ×4ms

|      |                                       |   |
|------|---------------------------------------|---|
| 0152 | 1Tool post expansion auxiliary time 1 | 0 |
|------|---------------------------------------|---|

Setting range: 0~9999999 Unit: ×4ms

|      |                                      |   |
|------|--------------------------------------|---|
| 0153 | Tool post expansion auxiliary time 2 | 0 |
|------|--------------------------------------|---|

Setting range: 0~9999999 Unit: ×4ms

|      |                                      |   |
|------|--------------------------------------|---|
| 0154 | Tool post expansion auxiliary time 3 | 0 |
|------|--------------------------------------|---|

Setting range: 0~9999999 Unit: ×4ms

|      |                                      |   |
|------|--------------------------------------|---|
| 0155 | Tool post expansion auxiliary time 4 | 0 |
|------|--------------------------------------|---|

Setting range: 0~9999999 Unit: ×4ms

|      |                           |   |
|------|---------------------------|---|
| 0156 | Pulse stationary constant | 0 |
|------|---------------------------|---|

Setting range: 0~9999999 Unit: ×4ms

|      |                                                |                     |
|------|------------------------------------------------|---------------------|
| 0157 | Linear ACC&DEC time constant in thread run-out | Step200<br>Servo 40 |
|------|------------------------------------------------|---------------------|

Setting range: 6~999 Unit: ×4ms

|      |                                   |                        |
|------|-----------------------------------|------------------------|
| 0158 | Upper speed of the thread run-out | Step6000<br>Servo 7600 |
|------|-----------------------------------|------------------------|

Setting range: 30~30000 Unit: ×4ms

|      |                                    |   |
|------|------------------------------------|---|
| 0159 | Additional alarm time of start key | 0 |
|------|------------------------------------|---|

Setting range: 0~9999999 Unit: ×4ms

Note: when it is set to be 0, system default to alarm after 3s by pressing start key, so the alarm time start point is 3s.

|      |             |   |
|------|-------------|---|
| 0160 | To be added | 0 |
|------|-------------|---|

Setting range: 0~9999999 Unit: ×4ms

|      |                                                   |   |
|------|---------------------------------------------------|---|
| 0161 | If Lubrication alarm is valid(0:invalid, 1:valid) | 0 |
|------|---------------------------------------------------|---|

Note: parameter 162~167 are to be added parameters.

|      |                    |   |
|------|--------------------|---|
| 0177 | CS axis type 1 2 3 | 0 |
|------|--------------------|---|

|      |                                    |   |
|------|------------------------------------|---|
| 0178 | CS check input point max. overtime | 0 |
|------|------------------------------------|---|

Setting range: 0~9999999 Unit:×4ms

|      |                          |   |
|------|--------------------------|---|
| 0179 | CS axis clamp delay time | 0 |
|------|--------------------------|---|

Setting range: 0~9999999 Unit:×4ms

|      |                                                                         |   |
|------|-------------------------------------------------------------------------|---|
| 0180 | CS axis positioning coordinate of Y axis, negative value do not input . | 0 |
|------|-------------------------------------------------------------------------|---|

Setting range: 0~9999999

|      |                                              |   |
|------|----------------------------------------------|---|
| 0181 | CS output delay time before torque reduction | 0 |
|------|----------------------------------------------|---|

Setting range: 0~9999999 Unit:×4ms: ×4ms

|      |                                       |   |
|------|---------------------------------------|---|
| 0182 | Interpolation period (0: 2MS; 1: 4MS) | 0 |
|------|---------------------------------------|---|

Setting range: 0~9999999 Unit:×4ms: ×4ms

|      |                            |   |
|------|----------------------------|---|
| 0183 | System type (1: TA; 2: TB) | 0 |
|------|----------------------------|---|

Setting range: 0~9999999 Unit:×4ms: ×4ms

|      |                                                        |   |
|------|--------------------------------------------------------|---|
| 0184 | +Hard limit input point of X axis(valid after restart) | 0 |
|------|--------------------------------------------------------|---|

Setting range: 0~9999999 Unit:×4ms: ×4ms

|      |                                                        |   |
|------|--------------------------------------------------------|---|
| 0185 | -Hard limit input point of X axis(valid after restart) | 0 |
|------|--------------------------------------------------------|---|

Setting range: 0~9999999 Unit:×4ms: ×4ms

|      |                                                        |   |
|------|--------------------------------------------------------|---|
| 0186 | +Hard limit input point of Z axis(valid after restart) | 0 |
|------|--------------------------------------------------------|---|

Setting range: 0~9999999 Unit:×4ms: ×4ms

|      |                                                        |   |
|------|--------------------------------------------------------|---|
| 0187 | -Hard limit input point of Z axis(valid after restart) | 0 |
|------|--------------------------------------------------------|---|

Setting range: 0~9999999 Unit:×4ms: ×4ms

|      |                                                                       |   |
|------|-----------------------------------------------------------------------|---|
| 0188 | 3 <sup>rd</sup> section speed of each axis zero (valid after restart) | 0 |
|------|-----------------------------------------------------------------------|---|

Setting range: 0~9999999 Unit:×4ms: ×4ms

|      |                                               |   |
|------|-----------------------------------------------|---|
| 0189 | Tail control type 0/1/2 (valid after restart) | 0 |
|------|-----------------------------------------------|---|

Setting range: 0~9999999 Unit:×4ms: ×4ms

|      |                                                |   |
|------|------------------------------------------------|---|
| 0190 | Truck control type 0/1/2 (valid after restart) | 0 |
|------|------------------------------------------------|---|

Setting range: 0~9999999 Unit:×4ms: ×4ms

|      |                                                            |   |
|------|------------------------------------------------------------|---|
| 0191 | Truck manual control allowable speed (valid after restart) | 0 |
|------|------------------------------------------------------------|---|

Setting range: 0~9999999 Unit:×4ms: ×4ms

Note: parameter 192~256 are to be added.

System parameter No.

|   |   |   |     |     |     |     |     |     |     |      |
|---|---|---|-----|-----|-----|-----|-----|-----|-----|------|
| 8 | 0 | 1 | SPI | SPC | CSL | CSP | CSZ | CSM | YCS | PEMG |
|---|---|---|-----|-----|-----|-----|-----|-----|-----|------|

**SPI** =1: spindle clamp input high-level valid

=0: spindle clamp input low-level valid

**SPC** =1: check spindle clamp input

=0: Do not check spindle clamp input

**CSL** =1: CS pos input high-level valid

=0: CS pos input low-level valid

**CSP** =1: CS position mode high-level valid

=0: CS position mode low-level valid

**CSZ** =1: CS zero speed high-level valid

=0: CS zero speed low-level valid

**CSM** =1: CS axis control M code type

=0: CS axis do not control M code type

**YCS** =1: Y axis set as CS axis

=0: Y axis do not set as CS axis

**PEMG** =1: MPG emergency stop signal high-level valid

=0: MPG emergency stop signal low-level valid

standard setting: 0 0 0 0 0 0 0 0

System parameter No.

|   |   |   |     |     |     |     |     |     |     |      |
|---|---|---|-----|-----|-----|-----|-----|-----|-----|------|
| 8 | 0 | 2 | TAR | TL1 | TL0 | TM1 | TM0 | TR1 | TR0 | ALMC |
|---|---|---|-----|-----|-----|-----|-----|-----|-----|------|

**TAR** =1: spindle alaem high-level valid

=0: spindle alaem low-level valid

**TL1** =1: spindle alaem high-level valid

=0: spindle alaem low-level valid

**TL0** =1: spindle alaem high-level valid

=0: spindle alaem low-level valid

**TM1** =1: spindle alaem high-level valid

=0: spindle alaem low-level valid

**TM0** =1: spindle alaem high-level valid

=0: spindle alaem low-level valid

**TR1** =1: spindle alaem high-level valid

=0: spindle alaem low-level valid

**TR0** =1: spindle alaem high-level valid

=0: spindle alaem low-level valid

**ALMC** =1: spindle alaem high-level valid

=0: spindle alaem low-level valid

standard setting: 0 0 0 0 0 0 0 0

System parameter No.

|   |   |   |  |  |  |  |  |     |     |     |
|---|---|---|--|--|--|--|--|-----|-----|-----|
| 8 | 0 | 3 |  |  |  |  |  | VMB | LTC | XZL |
|---|---|---|--|--|--|--|--|-----|-----|-----|

**VMB**      =1: spindle alarm high-level valid  
               =0: spindle alarm low-level valid

**LTC**      =1: spindle alarm high-level valid  
               =0: spindle alarm low-level valid

**XZL**      =1: spindle alarm high-level valid  
               =0: spindle alarm low-level valid

standard setting: 0 0 0 0    0 0 0 0

parameter 192~256 are to be added.

## Appendix II Alarm List

| No. | Content                                                                                 | Trouble shooting                                                                                                                  |
|-----|-----------------------------------------------------------------------------------------|-----------------------------------------------------------------------------------------------------------------------------------|
| 000 | Parameter for cutting off the primary power source is set, please cut off power supply. | Power on the CNC                                                                                                                  |
| 001 | Fail to open file.                                                                      | Open the right file.                                                                                                              |
| 002 | This character string can not be found in Edit mode                                     | Wrong operation, and search the existed character string again.                                                                   |
| 003 | Instruction pulse or speed is too big. Reset or tool setting is needed.                 | Set these parameters again: P21/P22/P23/P24. See parameter list and initialization disk of manual.                                |
| 004 | Address is not found                                                                    | Wrong operation, sequence number should be put before the program                                                                 |
| 005 | No data follows the address.                                                            | Modify the program. See programming manual for details.                                                                           |
| 006 | Negative sign entry error                                                               | Modify the program. See programming manual for details.                                                                           |
| 007 | Decimal point entry error                                                               | Modify the program. See programming manual for details.                                                                           |
| 008 | File creation is unsuccessful.                                                          | Delete the unused program, and then create a new file.                                                                            |
| 009 | Input address of the illegal word                                                       | Modify wrong address in the program. See programming manual for details.                                                          |
| 010 | Inactive G code is specified.                                                           | Modify wrong G code in the program. See programming manual for details.                                                           |
| 011 | F command value of cutting feed is wrong.                                               | Set cutting federate F again. Feedrate per minute: 1-8000mm/min, Feedrate per rev: 0.001-500mm/r                                  |
| 012 | Digits of input value exceed the maximum allowable range                                | Modify input data. See programming manual for details.                                                                            |
| 013 | The program specified the program number or the address to be searched are not found.   | Wrong operation, and search the existed program number again.                                                                     |
| 014 | The file to be deleted does not exist.                                                  | Delete the existed file again.                                                                                                    |
| 015 | Storage capacity of the memory is insufficient.                                         | Delete unneeded program                                                                                                           |
| 016 | In thread interpolation, spindle speed is low or spindle is not started.                | Start spindle when turning thread. Check whether the speed of the peripheral spindle is normal or whether the spindle is started. |
| 017 | Zero clearing is wrong                                                                  | Modify the program                                                                                                                |
| 018 | F,I command values are wrong in thread cutting.                                         | Set thread lead again, metric thread 0.001-500mm, inch thread 0.06-25400 tooth/inch.                                              |
| 019 | The length of the block exceeds 255 characters.                                         | Delete unneeded characters                                                                                                        |
| 020 | F value is not specified in tapping cycle.                                              | Lead F value is needed in tapping processing                                                                                      |
| 021 | In program zero return, zero return fails if there                                      | Firstly execute G50, and then perform program                                                                                     |

|     |                                                                                                                                                     |                                                                                                           |
|-----|-----------------------------------------------------------------------------------------------------------------------------------------------------|-----------------------------------------------------------------------------------------------------------|
|     | is no zero or zero point is at the edge of the block.                                                                                               | zero return. Adjust position of zero return block, and perform zero return again.                         |
| 022 | The factory setting value can not be covered.                                                                                                       | When operate illegally, press reset key to cancel the alarm.                                              |
| 023 | When circular interpolation defined by radius R is used, R defined a negative value.                                                                | Circular radius R should not be a negative value, modify the program.                                     |
| 024 | Radius of circular interpolation is too big.                                                                                                        | Modify radius R. See programming manual.                                                                  |
| 026 | The value specified in circular interpolation can not form a circular arc.                                                                          | Modify program, radius, starting coordinate value or ending coordinate values of the circular arc.        |
| 027 | In circular interpolation, R and (K,I) are specified simultaneously.                                                                                | Either R or (K,I) is used to specify radius of circular arc. Delete unneeded characters.                  |
| 028 | In circular interpolation, R and (K,I) are zero.                                                                                                    | Specify radius of circle correctly when checking program.                                                 |
| 029 | The offset value specified by T code is too big.                                                                                                    | Perform tool setting again, modify the offset value specified by T code.                                  |
| 030 | Tool offset number exceeds the range.                                                                                                               | Modify tool offset number, the range: 1-64                                                                |
| 031 | The file can not be found                                                                                                                           | Search the file again                                                                                     |
| 032 | Communication error                                                                                                                                 | Check whether the baud rate of CNC is consistent with that of the computer                                |
| 033 | There is no cross point in tool nose radius compensation                                                                                            | Modify the program                                                                                        |
| 034 | G2 or G3 is executed when starting or canceling the tool radius compensation.                                                                       | G2 or G3 can not be performed when starting or canceling the tool compensation. Modify the program.       |
| 035 | Compensation direction of the next block changes after the compensation is started.                                                                 | Tool compensation direction shall not be changed after the compensation is started. Modify the program.   |
| 036 | Tool radius compensation is performed before tool offset number is specified.                                                                       | Tool offset number shall be specified before executing tool compensation. Modify program, e.g. Add T0101. |
| 037 | Block is non-movinG Codewhen compensation starts.                                                                                                   | Block shall not be non-movinG Codewhen compensation starts.                                               |
| 038 | In tool nose radius compensation, excessive cutting occurs as starting point or terminal point of the circular arc coincides with the center point, | Modify the program                                                                                        |
| 039 | 30 non-movinG Codes appear continuously in the process of tool compensation.                                                                        | Delete unneeded blocks.                                                                                   |
| 040 | Illegal command appears in the process of tool compensation.                                                                                        | Check the program. See programming manual for details.                                                    |
| 041 | Excessive cutting will occur in tool nose radius compensation                                                                                       | Check the program. See programming manual for details.                                                    |
| 042 | In the process of tool nose radius compensation, G code is specified repeatedly in circular arc command.                                            | Modify the program. See programming manual for details.                                                   |
| 043 | Data input exceeds the allowable range.                                                                                                             | Modify data or check whether the destination address value exceeds the range.                             |

|     |                                                                                                                                                                                                                                                                                                                                                                             |                                                                                                                                                                                                                          |
|-----|-----------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|--------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| 044 | Spindle disabled if chuck is not clamped.                                                                                                                                                                                                                                                                                                                                   | Clamp the chuck before starting the spindle.                                                                                                                                                                             |
| 045 | Chuck unclamping is not allowed when spindle is in operation.                                                                                                                                                                                                                                                                                                               | Execute M05 before unclamping the chuck.                                                                                                                                                                                 |
| 046 | Storage is wrong at power off. Please power on the CNC.                                                                                                                                                                                                                                                                                                                     | Coordinate value is too big. Cancel the alarm by pressing reset key, and power on again.                                                                                                                                 |
| 047 | Spindle CW (rotates negatively) is specified before stopping spindle CCW (rotates positively).                                                                                                                                                                                                                                                                              | Perform M05, an then execute M03 or M04.                                                                                                                                                                                 |
| 050 | An unusable tool nose angle is specified in G76.                                                                                                                                                                                                                                                                                                                            | Modify the program, and specify a correct degree. See programming manual for details.                                                                                                                                    |
| 051 | In G76, minimum specified cutting depth or finishing allowance is bigger than thread height.                                                                                                                                                                                                                                                                                | Modify the program, and specify a correct cutting depth. See programming manual for details.                                                                                                                             |
| 052 | Thread height or initial cutting depth specified in G76, which is 0 or negative number.                                                                                                                                                                                                                                                                                     | Modify the program. See programming manual for details.                                                                                                                                                                  |
| 053 | P or Q value is not specified in G70, G71,G72,G73 code.                                                                                                                                                                                                                                                                                                                     | Modify the program. See programming manual for details.                                                                                                                                                                  |
| 054 | # I or # K in G74 or G75 is a negative value.                                                                                                                                                                                                                                                                                                                               | Modify the program. See programming manual for details.                                                                                                                                                                  |
| 055 | .<br>G76 finishing allowance is a negative value.                                                                                                                                                                                                                                                                                                                           | Range of finishing allowance: 0-9999.999 mm. Modify the program.                                                                                                                                                         |
| 062 | (1) Cutting depth in G71 or G72 is 0 or negative value.<br>(2) G73 repetition time is 0 or negative value<br>(3) D-i or D-k command in G74 or G75 is negative value<br>(4) While D-i or D-k command in G74 or G75 is not 0, address U or W is specified as 0 or negative value<br>(5) While tool retraction direction of G74 or G75 is specified, D-d is a negative number. | Modify the program. See programming manual.                                                                                                                                                                              |
| 063 | In G70, blocks specified by P or Q can not be found.                                                                                                                                                                                                                                                                                                                        | Modify the program, and add correct blocks.                                                                                                                                                                              |
| 064 | Tool compensation is not cancelled when G code is G32,G33,G90~G94,G70~G76.                                                                                                                                                                                                                                                                                                  | The above G codes can not be performed in tool compensation state. Execute G40 first.                                                                                                                                    |
| 065 | (1) In code G71,G72 or G73, in the block whose sequence number is specified by address P, G00 or G01 is not defined.<br>(2) In code G71 or G72, in the block whose sequence number is specified by address P, address Z(W)(G71) or X(U)(G72) are defined.                                                                                                                   | The first block in NS~NF should be G00 or G01.<br>In the first block of NS~NF specified by G71, X or U shall be defined separately.<br>In the first block of NS~NF specified by G72, Z or W shall be defined separately. |
| 066 | In code G70, G71,G72,G73, in the blocks specified by P or Q, non-movinG Codeor unusable G code is specified.                                                                                                                                                                                                                                                                | Modify the program. See programming manual for details.                                                                                                                                                                  |
| 067 | M98,M99 or M30.is specified in the blocks that is specified by address P or Q.                                                                                                                                                                                                                                                                                              | Modify the program and delete incorrect codes.                                                                                                                                                                           |
| 068 | P or Q value of G70 is specified repeatedly.                                                                                                                                                                                                                                                                                                                                | Modify the program. See programming manual for details.                                                                                                                                                                  |

|     |                                                                                                                                    |                                                                                                                        |
|-----|------------------------------------------------------------------------------------------------------------------------------------|------------------------------------------------------------------------------------------------------------------------|
| 074 | The program number is out of the range of 0001-9999.                                                                               | Modify the program.                                                                                                    |
| 076 | In the block M98, P is not specified.                                                                                              | Modify the program, and specify the correct P code.                                                                    |
| 077 | Subprogram calls too many nests.                                                                                                   | Called subprograms exceed the limit. Subprogram is a four-embedded program. Modify the program nest structure.         |
| 078 | The specified program number or sequence number has not been found in subprogram call.                                             | Modify the program, and check the program number or sequence number specified by P.                                    |
| 079 | Macro code alarm does not exist.                                                                                                   | Specify the correct macro code alarm number.                                                                           |
| 081 | NC program and customer macro program exist simultaneously.                                                                        | Modify the program. See programming manual for details.                                                                |
| 100 | Parameter switch is ON.                                                                                                            | Set parameter switch to the state of "OFF" (In MDI mode, press character "W" on the setting page), or press reset key. |
| 101 | In program editing, power is off. Please edit the program again.                                                                   | Press reset key to cancel the alarm, and edit the program again.                                                       |
| 102 | The number of lines exceed the maximum value.                                                                                      | Delete redundant lines.                                                                                                |
| 103 | Memory data of parameter, tool compensation and coordinate are initialized.                                                        | Read initialized disk again. If they still exist after power on, please change the main board.                         |
| 104 | Because CNC abnormality is detected, it needs to check parameter again.                                                            | Read initialized disk again. If they still exist after power on, please change the main board.                         |
| 105 | It detects data of tool length compensation is abnormal. Please set the tool again.                                                | Read initialized disk again. If they still exist after power on, please change the main board.                         |
| 106 | It detects data of the absolute data is abnormal. Please set tool again.                                                           | Read initialized disk again. If they still exist after power on, please change the main board.                         |
| 107 | It detects the drive, spindle etc. are changing.                                                                                   | Restart the system again.                                                                                              |
| 110 | When copied and renamed, the program name already exists                                                                           | Select a new program name, and perform again.                                                                          |
| 114 | In G65 block, undefined H code is specified.                                                                                       | Check program, and specify the correct H code.                                                                         |
| 115 | Illegal variable number is specified.                                                                                              | Modify the program, and specify the correct variable number.                                                           |
| 128 | When transferring the command, the sequence number of address to be transferred is not in the allowable range, or it is not found. | Modify the program, and use the correct sequence number.                                                               |
| 129 | A block must not contain identical function words or same series of G code.                                                        | Modify the program. See programming manual for details.                                                                |
| 130 | The value of the address exceeds the range.                                                                                        | There are specified ranges for addresses. Modify the program. See programming manual for details.                      |
| 131 | Group G00 and group G01 must not appear at the same time.                                                                          | Modify the program, and delete redundant G code.                                                                       |

|     |                                                                                                                                                                        |                                                                                                            |
|-----|------------------------------------------------------------------------------------------------------------------------------------------------------------------------|------------------------------------------------------------------------------------------------------------|
| 132 | Coordinate value X and U or Z and W appear at the same time.                                                                                                           | Modify the program, and delete redundant coordinate value.                                                 |
| 133 | Two repeatable canned cycle G code are specified continuously.                                                                                                         | Modify the program. See the programming manual for details.                                                |
| 134 | When specifying G32, G33, G92, I, F appear at the same time.                                                                                                           | Only F or I specify the screw pitch. Delete unwanted characters.                                           |
| 137 | Too many blocks in repeatable canned cycle G71 or G72, G73.                                                                                                            | Number of block shall not exceed 100. Delete unwanted blocks.                                              |
| 138 | N must be increased progressively.                                                                                                                                     | Modify sequence number N. It must be increased progressively.                                              |
| 140 | There must be a space between addresses.                                                                                                                               | Modify the program. See programming manual for details.                                                    |
| 141 | Constant speed command is specified when the spindle uses gear control.                                                                                                | When using constant speed, spindle shall use frequency conversion control. Change parameter NO.1Bit4 to 1. |
| 142 | Program ends illegally without M30 or M99.                                                                                                                             | Modify the program. See programming manual for details.                                                    |
| 144 | Infeed value, retraction value, chamfer allowance, finishing times or teeth height of the screw cutting in G70, G71, G72, G73, G74, G75, G76 are wrong.                | Modify the program. See programming manual for details.                                                    |
| 145 | The value of address in G70, G71, G72, G73, G74, G75, G76                                                                                                              | Modify the program. See programming manual for details.                                                    |
| 146 | Variable shall not be used by address O and N.                                                                                                                         | Address O and N should not be used for macro variable. Modify the program.                                 |
| 147 | P in G65 should not be a constant.                                                                                                                                     | Modify the program. See programming manual for details.                                                    |
| 149 | Tool radius compensation is not cancelled before entering or exiting the subprogram.                                                                                   | Tool compensation should be cancelled before calling the subprogram. Add G40.                              |
| 150 | Some coordinates are not changed monotonously in repeatable canned cycle.                                                                                              | Check the program. See the programming manual for details.                                                 |
| 151 | U and W in G71 and G72 code are not consistent with the track path, or U(W) do not exist or the values exceed the range. The coordinates are not changed monotonously. | Check the program. See the programming manual for details.                                                 |
| 152 | Exceed the circumstances that G76,G90,G92,G94can process.                                                                                                              | Check the program. See the programming manual for details.                                                 |
| 153 | F, I values in G33, G92 codes are wrong.                                                                                                                               | Set thread lead again, metric thread 0.001-500mm, inch thread 0.06-25400 tooth/inch.                       |
| 155 | Constant surface speed command is wrong.                                                                                                                               | Check the program. See the programming manual for details.                                                 |
| 156 | In G71, G72, G73, the blocks specified by P are not closed to each other. G71, G72, G73 blocks or the blocks specified by P or Q do not exist.                         | Check the program. See the programming manual for details.                                                 |
| 158 | G70,G71,G72 or the P value specified by G73 are bigger than or equal to Q value.                                                                                       | P must be bigger than Q. Modify the program.                                                               |

|     |                                                                                                                                                    |                                                                                                                                                                                                                          |
|-----|----------------------------------------------------------------------------------------------------------------------------------------------------|--------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| 162 | P is non-integral value or a negative value in the blocks of the called program.                                                                   | Check the program. See the programming manual for details.                                                                                                                                                               |
| 163 | Subprogram is called too many times.                                                                                                               | Subprogram call times: 1-999. Modify the program.                                                                                                                                                                        |
| 164 | Exceed the maximum storage number (500)                                                                                                            | Delete unnecessary program.                                                                                                                                                                                              |
| 165 | The sentence for calling subprogram or returning subprogram is in illegal block (G90-G94, G70~G76).                                                | Check the program. See the programming manual for details.                                                                                                                                                               |
| 166 | Pno and Qno values assigned by G70, G71, G72, and G73 exceed the range or they are non-integral.                                                   | Check the program. See the programming manual for details.                                                                                                                                                               |
| 168 | P,U and X in the pause command can not be specified at the same time.                                                                              | Delete unnecessary address characters.                                                                                                                                                                                   |
| 169 | Q and R in macro program are specified incorrectly.                                                                                                | Check the program. See the programming manual for details.                                                                                                                                                               |
| 170 | Please input correct password in setting interface before modifying parameter and screw pitch compensation data.                                   | It is necessary to input password before modifying parameter and screw pitch. See programming manual for details.                                                                                                        |
| 171 | Spindle enable signal is not received.                                                                                                             | Check SPEN input state of parameter NO.3, and check whether circuit of the external spindle enable signal is normal, or change parameter NO.15 Bit1 to 0: detection for spindle enable signal is shielded.               |
| 180 | M41~M44 can not be used when spindle is in gear control or spindle automatic gearing is inactive.                                                  | Requirements for using M41-M44: frequency conversion and spindle automatic gearing is active. Change parameter NO.1 Bit4 and NO.10 Bit7 to 1.                                                                            |
| 181 | M code is wrong. Illegal M code is edited in the program.                                                                                          | Corresponding parameter is not set. See parameter list of the manual.                                                                                                                                                    |
| 182 | S code is wrong. Illegal S code is edited in the program.                                                                                          | Corresponding parameter is not set. See parameter list of the manual.                                                                                                                                                    |
| 183 | T code is wrong. Illegal T code is edited in the program.                                                                                          | Corresponding parameter is not set. See parameter list of the manual.                                                                                                                                                    |
| 185 | Time for tool setting is too long. Specified tool arrival signal has not been received Ta time after the start of tool CCW rotation. Alarm occurs. | According to input state of NO.2 T01-T08, check whether the setting of NO.9 Bit1 is correct or adjust P82 tool changing time.                                                                                            |
| 186 | Tool post CW rotation locking signal is not received in the time of tool post CW rotation locking time.                                            | According to TCP input state of parameter NO.1 Bit7, check whether the setting of NO.9 Bit0 is correct                                                                                                                   |
| 187 | Tool changing is not ready. Tool number is not consistent with the current tool number.                                                            | Press reset key to cancel the alarm or adjust position of signal sending disk, or according to the state of NO.2 T01-T08, adjust NO.9 Bit1 level setting or change NO.10 Bit2 to 0: Shield tool number detection signal. |
| 188 | M45--M49 commands are executed while corresponding in-position input signal is not received.                                                       | In-position input signal is necessary for executing M45-M49. Check whether the circuit is correct.                                                                                                                       |

|     |                                                                                                                                     |                                                                                                                                                                                                       |
|-----|-------------------------------------------------------------------------------------------------------------------------------------|-------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| 189 | Chuck in-position signal is not received within the stipulated time.                                                                | Chuck in-position detection is active. Check whether the wire of external alarm is correct. Adjust parameter P91 arrival time or modify parameter NO.15 Bit2 to 0: Shield chuck in-position.          |
| 190 | Axis number is changed, and data initialization is successful. After turning off the parameter switch, press reset key to continue. | Press reset key to cancel the alarm when CNC alarms.                                                                                                                                                  |
| 191 | Key pressing is overtime.                                                                                                           | It is not allowed to press cycle start key continuously. Adjust parameter P159 key pressing time.                                                                                                     |
| 193 | Please pull out U-disk!                                                                                                             | Operation is wrong. U-disk should not be plugged in during processing.                                                                                                                                |
| 201 | Exceed X-axis positive stroke limit.                                                                                                | Modify the range of X-axis positive stroke of the CNC parameter, or cancel alarm by manual negative feed.                                                                                             |
| 202 | Exceed X-axis negative stroke limit.                                                                                                | Modify the range of X-axis negative stroke of the CNC parameter or cancel alarm by manual negative feed.                                                                                              |
| 203 | Exceed X-axis positive stroke limit.                                                                                                | Modify the range of Z-axis positive stroke of the CNC parameter or cancel alarm by manual negative feed.                                                                                              |
| 204 | Exceed Z-axis negative stroke limit.                                                                                                | Modify the range of Z-axis negative stroke of the CNC parameter or cancel alarm by manual negative feed.                                                                                              |
| 205 | Exceed Y-axis positive stroke limit.                                                                                                | Modify the range of Y-axis positive stroke of the CNC parameter or cancel alarm by manual negative feed.                                                                                              |
| 206 | Exceed Y-axis negative stroke limit.                                                                                                | Modify the range of Y-axis negative stroke of the CNC parameter or cancel alarm by manual negative feed.                                                                                              |
| 207 | X-axis hardware overtravel (negative).                                                                                              | When hardware detection is active, check whether the external alarm circuit is correct, or change alarm level parameter NO.8 Bit4, Bit5 or parameter NO.8 Bit2, Bit3 to 0: Shield hardware detection. |
| 208 | Z-axis hardware overtravel (negative).                                                                                              |                                                                                                                                                                                                       |
| 209 | X-axis hardware overtravel (positive).                                                                                              |                                                                                                                                                                                                       |
| 210 | Z-axis hardware overtravel (positive).                                                                                              |                                                                                                                                                                                                       |
| 211 | X-axis drive unit is not ready.                                                                                                     | Level of the alarm signal is not correct. Adjust CNC parameter NO.8 again. Check whether the drive unit is damaged.                                                                                   |
| 212 | Z-axis drive unit is not ready.                                                                                                     |                                                                                                                                                                                                       |
| 213 | X/Z -axis drive unit is not ready.                                                                                                  |                                                                                                                                                                                                       |
| 221 | X-axis drive unit alarm.                                                                                                            | Level of the alarm signal is not correct. Adjust CNC parameter NO.8 Bit0, Bit1. Check whether the drive unit is damaged.                                                                              |
| 222 | Z-axis drive unit alarm.                                                                                                            |                                                                                                                                                                                                       |
| 223 | X-axis overtravel alarm.                                                                                                            |                                                                                                                                                                                                       |
| 224 | Z-axis overtravel alarm.                                                                                                            |                                                                                                                                                                                                       |
| 250 | Tailstock advance/retreat can not be performed in Auto mode or after the spindle is started.                                        | Operation is wrong. Execute M05 and then perform M10/M11.                                                                                                                                             |
| 251 | External alarm 1.                                                                                                                   | External alarm 1 is active. Check whether the external alarm circuit is correct, or change alarm                                                                                                      |

|     |                                                                                                                                                      |                                                                                                                                                                               |
|-----|------------------------------------------------------------------------------------------------------------------------------------------------------|-------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
|     |                                                                                                                                                      | level parameter NO.12 Bit2 or NO.12 Bit6 to 0: Shield pressure detection.                                                                                                     |
| 252 | External alarm 2.                                                                                                                                    | External alarm 2 is active. Check whether the external alarm circuit is correct, or change alarm level parameter NO.12 Bit3 or NO.12 Bit7 to 0: Shield pressure detection.    |
| 253 | Pressure is too low.                                                                                                                                 | Pressure detection is active. Check whether the pressure circuit is correct, or change alarm level parameter NO.16 Bit3 or NO.16 Bit4 to 0: Shield pressure detection.        |
| 254 | Preparation is not ready.                                                                                                                            | CNC is in emergency stop state. Check whether the emergency stop button is pressed down, or change parameter NO.14 Bit3 to 1: Shield emergency stop.                          |
| 255 | Safety door is open in automatic operation.                                                                                                          | Operation is wrong. Safety door should not be opened during the operation. Change alarm level parameter NO.16 Bit6 or parameter NO.16 Bit5 to 0: Shield safety door function. |
| 258 | IO port setting is wrong. It needs to be set again in PLC-V. Wrong setting of IO port will cause accident. Please contact the manufacturer for help. | Main board has faults. Please contact the manufacturer.                                                                                                                       |
| 259 | Timer has faults. Please contact the manufacturer for help.                                                                                          | Main board has faults. Please contact the manufacturer.                                                                                                                       |

## Appendix III Diagnosis List

### 1. DI/DO Diagnosis Message

#### 1.1 Input Signal from Machine Side (I/O board)

Input signal diagnosis message from the machine side directly corresponds to CNC hardware circuit. If there is a fault on the hardware circuit, content described in this chapter can be used for confirmation. For the specific position of signals on PCB (printed circuit board), please refer to logic diagram.

##### 1) Diagnosis list:

| Diagnosis No. |   |   | Significance |      |       |      |      |      |      |      |
|---------------|---|---|--------------|------|-------|------|------|------|------|------|
| 0             | 0 | 1 | *TCP         | DIQP | *DECX | DITW | *SP  | ST   | DECZ | *ESP |
| 0             | 0 | 2 | T08          | T07  | T06   | T05  | T04  | T03  | T02  | T01  |
| 0             | 0 | 3 | SAR          | LTZ  | LTX   | SPEN | PCH  | DOOR | GR2  | GR1  |
| 0             | 0 | 4 | M01          | M93  | M91   | LTY  | DECY | LCK  | OWA2 | OWA1 |

##### 2) Significance of the signal

- (a) \*TCP: Tool post clamping signal
- (b) DIQP: Chuck foot switch (input signal, input is active when it connects to +24V)
- (c) DITW: Tailstock input signal
- (d) \*DECX. \*DECZ: Deceleration signal for corresponding axis
- (e) \*SP: Feed hold signal
- (f) ST: Automatic cycle start key
- (g) \*ESP: Emergency stop signal
- (h) T01~T08: Tool post in-position signal
- (i) SAR: Spindle speed in-position signal
- (j) LTZ, LTX: Stroke limit signal for corresponding axis
- (k) SPEN: Spindle enable signal
- (l) PCH: Pressure check signal in cycle start
- (m) DOOR: Safety door signal
- (n) GR1. GR2: Gear 1, 2 shifting in-position signal
- (o) M01: Conditional pause signal input port
- (p) M91 . M93: Two user-defined input ports
- (q) LTY: Stroke limit signal for corresponding axis
- (r) DECY: Zero point deceleration signal for corresponding axis
- (s) LCK: Keyboard locking signal (temporarily inactive)
- (t) OWA1. OWA2: External alarm 1, 2 signal

#### 1.2 Signal Input to Machine Side (I/O board)

When the corresponding bit is “1”, its corresponding output transistor in the hardware circuit is connected. If not, it means malfunction in output circuit. For its relationship with hardware, please refer to logic diagram.

## 1) Diagnosis

Diagnosis No.

Significance

|   |   |   |
|---|---|---|
| 0 | 0 | 5 |
|---|---|---|

|      |       |    |     |    |     |    |    |
|------|-------|----|-----|----|-----|----|----|
| SPZD | DOQPJ | M5 | M32 | M8 | M10 | M4 | M3 |
|------|-------|----|-----|----|-----|----|----|

|   |   |   |
|---|---|---|
| 0 | 0 | 6 |
|---|---|---|

|     |     |       |     |     |     |     |     |
|-----|-----|-------|-----|-----|-----|-----|-----|
| TL- | TL+ | DOQPS | M11 | M44 | M43 | M42 | M41 |
|-----|-----|-------|-----|-----|-----|-----|-----|

|   |   |   |
|---|---|---|
| 0 | 0 | 7 |
|---|---|---|

|     |     |     |      |       |     |     |     |
|-----|-----|-----|------|-------|-----|-----|-----|
| M53 | M51 | TLC | STAR | TRIAN | WAR | M30 | MST |
|-----|-----|-----|------|-------|-----|-----|-----|

|   |   |   |
|---|---|---|
| 0 | 0 | 8 |
|---|---|---|

|     |     |     |     |     |     |     |     |
|-----|-----|-----|-----|-----|-----|-----|-----|
| M69 | M67 | M65 | M63 | M61 | M59 | M57 | M55 |
|-----|-----|-----|-----|-----|-----|-----|-----|

## 2) Significance of the signal

- (a) SPZD: Spindle brake
- (b) DOQPJ: Chuck clamp output signal
- (c) M5: Spindle stop
- (d) M32: Lubrication ON
- (e) M8: Cooling ON
- (f) M10: Tailstock advance
- (g) M4: Spindle CW
- (h) M3: Spindle CCW
- (i) TL+, TL-: Tool changing signal
- (j) TL+ is positive tool changing signal. TL- is negative tool changing signal.
- (k) DOQPS; Chuck releasing output signal
- (l) M11: Tailstock retreat
- (m) M41~M44: Select spindle frequency conversion control. Gearing output in automatic gear shifting.
- (n) STAR. TRIAN: Machine start mode output control (star triangle starts)
- (o) WAR: Alarm signal
- (p) M30: Program end signal
- (q) MST: Automatic run signal
- (r) M51~M69; User-defined output port

## 2. CNC Interface Signal

## 1) Diagnosis list

Diagnosis No.

Significance

|   |   |   |
|---|---|---|
| 0 | 0 | 9 |
|---|---|---|

|  |  |     |     |  |     |     |     |
|--|--|-----|-----|--|-----|-----|-----|
|  |  | RFZ | RFR |  | PCS | PCZ | PCX |
|--|--|-----|-----|--|-----|-----|-----|

|   |   |   |
|---|---|---|
| 0 | 1 | 0 |
|---|---|---|

|  |  |  |  |  |  |      |      |
|--|--|--|--|--|--|------|------|
|  |  |  |  |  |  | ALMZ | ALMX |
|--|--|--|--|--|--|------|------|

|   |   |   |
|---|---|---|
| 0 | 1 | 1 |
|---|---|---|

MPG data

|   |   |   |
|---|---|---|
| 0 | 1 | 2 |
|---|---|---|

Spindle feedback data

|   |   |   |
|---|---|---|
| 0 | 1 | 3 |
|---|---|---|

Spindle feedback data

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

Spindle analog output value

|   |   |   |
|---|---|---|
| 0 | 1 | 5 |
|---|---|---|

Spindle analog output value

|   |   |   |
|---|---|---|
| 0 | 1 | 6 |
|---|---|---|

|  |       |  |  |     |     |     |     |
|--|-------|--|--|-----|-----|-----|-----|
|  | *MZRO |  |  | TYD | TYC | TYB | TYA |
|--|-------|--|--|-----|-----|-----|-----|

## 2) Signal significance

- (a) PCS: Spindle encoder one turn signal
- (b) PCX. PCZ: Zero point signal for corresponding axis
- (c) ALMX. ALMZ: Drive unit alarm for corresponding axis

These two diagnosis signals lie in ALM of the hardware circuit, and X stands for x axis, Z stands for z axis. №008 ALMZ, ALMX relate to the active level of this signal. E.g. If set No.8 ALMX to 1, drive unit of corresponding axis will alarm when ALMX is set to 0. On the contrary, corresponding axis will alarm when ALMX is set to 1. If the drive unit you used can not provide this signal, please do not connect this signal when wiring. In addition, №008 ALMZ, ALMX shall be set to 0. When drive unit alarms, please judge troubles at CNC side or at drive unit.

## (d) 011: MPG data

MPG data are mainly used for diagnosing the total number of pulse sent from X-axis or Z-axis to the motor in MPG mode. MPG speed can be seen from the speed of data changing.

## (e) Spindle feedback data

Number 012 and 013 display low-order 2 digits and high-order 2 digits of the spindle current speed in the form of BCD code.

## (f) Spindle analog output value

After spindle speed is defined, it is changed to the speed of controlling analog voltage output through gear and conversion control. Diagnosis number 013 and 014 display low-order 2 digits and high-order 2 digits of the speed after frequency conversion in the form of BCD code.

## 3) No alarm issues, and the motor does not move

- (a) CNC position display does not change  
Check whether the CNC message is correct.
- (b) CNC position display changes

First, check whether the CNC is in machine lock mode. If it is in this mode, machine coordinate display will not change. Ensure the display is changing.

## 3. Operation Panel Key

№017~022. 027~028: machine operation panel key diagnosis.

№023~026: keyboard key diagnosis

When the cursor under these diagnosis numbers, details for the corresponding keys are displayed at the lower part of the LCD. Press down the key on the operation panel, corresponding digit will display “1”, and it displays “0” after releasing. Otherwise, it indicates fault occurs.

## 4. CNC Input/Output Signal

Signals described in this part are signals transmitted between PMC of internal CNC and CNC. They help users to understand CNC internal working state, which is not relevant to hardware circuit directly.

Signals to CNC

Diagnosis No.

Significance

|   |   |   |
|---|---|---|
| 0 | 3 | 3 |
|   |   |   |

|                             |  |                        |  |              |              |  |  |
|-----------------------------|--|------------------------|--|--------------|--------------|--|--|
| <b>HX/RV1</b>               |  | <b>*DECX</b>           |  | <b>-X</b>    | <b>+X</b>    |  |  |
| MPGX<br>Rapid<br>override 1 |  | X-axis<br>decelerateon |  | Manual<br>-X | Manual<br>+X |  |  |

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

|                          |  |                        |  |              |              |  |  |
|--------------------------|--|------------------------|--|--------------|--------------|--|--|
| <b>HX/RV2</b>            |  | <b>*DECZ</b>           |  | <b>-Z</b>    | <b>+Z</b>    |  |  |
| MPGX Rapid<br>override 1 |  | X-axis<br>deceleration |  | Manual<br>-Z | Manual<br>+Z |  |  |

|   |   |   |
|---|---|---|
| 0 | 3 | 5 |
|   |   |   |

|            |  |  |  |                              |                              |  |  |
|------------|--|--|--|------------------------------|------------------------------|--|--|
| <b>DRN</b> |  |  |  | <b>GR2</b>                   | <b>GR1</b>                   |  |  |
| Dry run    |  |  |  | Gear<br>shifting<br>signal 2 | Gear<br>shifting<br>signal 1 |  |  |

|   |   |   |
|---|---|---|
| 0 | 3 | 6 |
|   |   |   |

|                 |                                  |            |            |              |            |  |  |
|-----------------|----------------------------------|------------|------------|--------------|------------|--|--|
| <b>MLK</b>      | <b>MP2</b>                       | <b>MP1</b> | <b>MP0</b> | <b>SBK</b>   | <b>BDT</b> |  |  |
| Machine<br>lock | Handwheel increment<br>selection |            |            | Single block | Block skip |  |  |

|   |   |   |
|---|---|---|
| 0 | 3 | 7 |
|   |   |   |

|                          |                 |                                      |                                 |                           |                |                     |            |
|--------------------------|-----------------|--------------------------------------|---------------------------------|---------------------------|----------------|---------------------|------------|
| <b>ZRN</b>               | <b>*SSTP</b>    | <b>SOR</b>                           | <b>SAR</b>                      | <b>FIN</b>                | <b>ST</b>      | <b>STLK</b>         | <b>MIX</b> |
| Zero<br>return<br>switch | Spindle<br>stop | Spindle<br>orientation<br>or gearing | Spindle<br>speed<br>in-position | MST<br>function<br>finish | Cycle<br>start | Interlock<br>signal | X-axis     |

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

|                   |                                   |            |                   |  |  |  |  |
|-------------------|-----------------------------------|------------|-------------------|--|--|--|--|
| <b>ERS</b>        | <b>RT</b>                         | <b>*SP</b> | <b>*ESP</b>       |  |  |  |  |
| External<br>reset | Manual<br>rapid feed<br>selection | Feed hold  | Emergency<br>stop |  |  |  |  |

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

|                                                        |            |            |            |                                                               |            |            |            |
|--------------------------------------------------------|------------|------------|------------|---------------------------------------------------------------|------------|------------|------------|
| <b>PN8</b>                                             | <b>PN4</b> | <b>PN2</b> | <b>PN1</b> | <b>KEY</b>                                                    | <b>MD4</b> | <b>MD2</b> | <b>MD1</b> |
| PN8~PN1:<br>External<br>program<br>number<br>selection |            |            |            | KEY: Program protection key/switch<br>MD4~MD1: Mode selection |            |            |            |

## 5. Signals from CNC

### 1) Diagnosis list

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

|           |           |            |            |            |  |            |            |
|-----------|-----------|------------|------------|------------|--|------------|------------|
| <b>OP</b> | <b>SA</b> | <b>STL</b> | <b>SPL</b> | <b>ENB</b> |  | <b>ZPZ</b> | <b>ZPX</b> |
|-----------|-----------|------------|------------|------------|--|------------|------------|

|   |   |   |
|---|---|---|
| 0 | 4 | 2 |
|---|---|---|

|           |  |  |  |            |  |            |           |
|-----------|--|--|--|------------|--|------------|-----------|
| <b>MA</b> |  |  |  | <b>DEN</b> |  | <b>RST</b> | <b>AL</b> |
|-----------|--|--|--|------------|--|------------|-----------|

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

|  |  |            |  |           |           |  |           |
|--|--|------------|--|-----------|-----------|--|-----------|
|  |  | <b>DST</b> |  | <b>TF</b> | <b>SF</b> |  | <b>MF</b> |
|--|--|------------|--|-----------|-----------|--|-----------|

|   |   |   |
|---|---|---|
| 0 | 4 | 4 |
|---|---|---|

|            |            |            |            |            |            |            |            |
|------------|------------|------------|------------|------------|------------|------------|------------|
| <b>M28</b> | <b>M24</b> | <b>M22</b> | <b>M21</b> | <b>M18</b> | <b>M14</b> | <b>M12</b> | <b>M11</b> |
|------------|------------|------------|------------|------------|------------|------------|------------|

|   |   |   |     |     |     |     |     |     |     |     |
|---|---|---|-----|-----|-----|-----|-----|-----|-----|-----|
| 0 | 4 | 5 | S28 | S24 | S22 | S21 | S18 | S14 | S12 | S11 |
| 0 | 4 | 6 | S48 | S44 | S42 | S41 | S38 | S34 | S32 | S31 |
| 0 | 4 | 9 | T8  | T7  | T6  | T5  | T4  | T3  | T2  | T1  |

## 2) Signal significance

- (a) OP: In auto running
- (b) SA: Servo ready
- (c) STL: Cycle start
- (d) SPL: Feed hold
- (e) ENB: Spindle enable
- (f) ZPX, ZPZ: X, Z axis zero return finish signal
- (g) MA: CNC ready signal
- (h) DEN: Axis move end signal
- (i) RST: Reset signal
- (j) AL: Alarm signal
- (k) DST: MDI start
- (l) TF, SF, MF: M, S, T code strobe signal
- (m) No. 044: BCD code of the current M code  
No. 045: Low-order 2 BCD codes of the current S code  
No. 046: High-order 2 BCD codes of the current S code  
No. 049: BCD code of the current T code

## 6. CNC Internal State Message

Current CNC working state can be viewed by No. 096 and No. 097 diagnosis messages if there is no alarm and movement during CNC auto running.

|   |   |   |  |      |      |      |      |       |      |      |
|---|---|---|--|------|------|------|------|-------|------|------|
| 0 | 5 | 0 |  | CSCT | CITL | COVL | CINP | CDVVL | CMTN | CFIN |
|---|---|---|--|------|------|------|------|-------|------|------|

When corresponding display is “1”, its significance as follows:

- CFIN: M, S, T code is being executed
- CMTN: Move code is being executed
- CDWL: G04 (pause) code is being executed
- CINP: Doing bit check
- COVZ: Override is 0%
- CITL: Interlock signal is ON
- CSCT: Waiting for SAR signal

|   |   |   |  |  |  |      |  |  |      |      |
|---|---|---|--|--|--|------|--|--|------|------|
| 0 | 5 | 1 |  |  |  | CRST |  |  | CTRD | CTPU |
|---|---|---|--|--|--|------|--|--|------|------|

When corresponding display is “1”, its significance as follows:

- CTPU: RS232 interface is transmitting
- CTRD: RS232 interface is receiving
- CTST: Emergency stop, external reset or reset key on the MDT panel is ON

|   |   |   |
|---|---|---|
| 0 | 5 | 2 |
|---|---|---|

|     |      |     |  |      |  |  |     |
|-----|------|-----|--|------|--|--|-----|
| STP | REST | EMS |  | RSTB |  |  | CSU |
|-----|------|-----|--|------|--|--|-----|

This diagnosis number is for the state when auto run stops. It is used for checking source of the trouble.

STP: Interpolation assignment stops, it is set in the following cases:

- 1) External reset is ON
- 2) Emergency stop is ON
- 3) Feed hold is On
- 4) Reset key on the MDI panel is ON
- 5) In Auto run, the mode is changed to Manual mode

REST: External reset key, reset key, emergency stop is ON

EMS: Emergency stop is ON

RSTB: Reset key on the MDI panel is ON

CSU: Emergency stop/ servo alarm active

|   |   |   |
|---|---|---|
| 0 | 5 | 3 |
|---|---|---|

Pulse sent to servo unit from (X) axis

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

Pulse sent to servo unit from (Z) axis

Pulses sent to servo unit are pulses from X or Z axis, which can be positive or negative.

## Appendix IV Specification List

### ● Functions

| Name                                | Specification                                                        |
|-------------------------------------|----------------------------------------------------------------------|
| Controlled axes                     | 2-axis (X, Z)                                                        |
| Simultaneous controlled axes        | 2-axes                                                               |
| Minimum setting unit                | 0.001mm                                                              |
| Minimum travel unit                 | 0.001mm                                                              |
| Maximum command                     | ±7 bits                                                              |
| Rapid traverse speed                | Standard: 7.6m/min, max. speed: 30 m/min                             |
| Rapid override                      | F0. 25%. 50%. 100%                                                   |
| Feed per minute                     | 1-8000mm/min                                                         |
| Maximum spindle speed               | 9999                                                                 |
| Thread lead                         | Metric: 0.001 mm~500.000mm<br>Inch: 0.06 teeth/inch~25400 teeth/inch |
| Automatic acceleration/deceleration | Available                                                            |
| Feedrate override                   | 0%~150% 16-gear in total                                             |
| Manual continuous feed              | 2-axis simultaneously                                                |
| Interpolation                       | Linear/circular                                                      |
| Reference return                    | Available                                                            |
| Display                             | TFT color LCD 480×240 or blue LCD 320×240                            |
| Step feed                           | 0.001mm. 0.01mm. 0.1mm                                               |
| MPG                                 | 0.001mm. 0.01mm. 0.1mm                                               |
| I/O interface                       | RS232C                                                               |
| Pause(s)                            | Available                                                            |
| Machine lock                        | All axes                                                             |
| Storage stroke check                | Available                                                            |
| Exact stop                          | Available                                                            |
| Reference return when power off     | Available                                                            |
| MDI operation                       | Available                                                            |
| Reset                               | Available                                                            |
| Dry run                             | Available                                                            |
| Single block                        | Available                                                            |
| Self diagnosis function             | Available                                                            |
| Emergency stop                      | Available                                                            |

### ● Display

| Name                                    | Specification |
|-----------------------------------------|---------------|
| Display                                 | Chinese       |
| Time, machined workpiece number display | Available     |
| Actual speed display                    | Available     |

- Program input

| Content                   | Specification |
|---------------------------|---------------|
| Coordinate system setting | Available     |
| Decimal point input       | Available     |
| User macro program        | Available     |
| Tool compensation         | Available     |
| Canned cycle              | Available     |
| Backlash compensation     | Available     |
| Radius R specification    | Available     |

- M. S. T function

| Name         | Specification |
|--------------|---------------|
| MST function | M 2-digit     |
| MST lock     | Available     |

- Spindle function

| Name                  | Specification         |
|-----------------------|-----------------------|
| Spindle function      | S 2-digit @ S 4-digit |
| Spindle analog output | Available             |
| Spindle override      | 50%~120%              |

- Tool function

| Name                           | Specification       |
|--------------------------------|---------------------|
| Tool function                  | T01~T64             |
| Tool compensation storage unit | ±6 digits 64 groups |
| Tool length compensation       | Available           |

- Edit, Operation

| Name                          | specification |
|-------------------------------|---------------|
| Program storage capacity      | 24544KB       |
| Number of the saved programs  | 500           |
| Display of the program number | Available     |
| Sequence number search        | Available     |
| Program number search         | Available     |
| Program protection            | Available     |

## Appendix V Wiring for CNC Matching Taiwan LIO SHING Tool Post

When GSK980TB3\TB3-V is used with Taiwan LIO SHING tool post, change parameter P81 to 6, P152 to 5, and P148 to 2.

|      |    |       |  |                                   |
|------|----|-------|--|-----------------------------------|
| XS40 | 23 | + 24V |  | + 24V                             |
|      | 12 | TL +  |  | Tool post CCW magnetic valve      |
|      | 13 | TL -  |  | Tool post CW magnetic valve       |
|      | 6  | T1    |  | Sensor A                          |
|      | 5  | T2    |  | Sensor B                          |
|      | 4  | T3    |  | Sensor C                          |
|      | 3  | T4    |  | Sensor D                          |
|      | 19 | T8    |  | Sensor E                          |
| XS39 | 12 | TCP   |  | Sensor F                          |
| XS42 | 16 | TLOCK |  | Tool post push-off magnetic valve |
|      | 25 | + 24V |  | + 24V                             |
|      | 24 | AGND  |  | 0V                                |

|                         |                                            |
|-------------------------|--------------------------------------------|
| <b>TCP (XS39: 12)</b>   | Tool post clamping signal                  |
| <b>+24V (XS40: 23)</b>  | +24V                                       |
| <b>TL+ (XS40: 12)</b>   | Tool post CCW signal                       |
| <b>TL- (XS40: 13)</b>   | Tool post CW signal                        |
| <b>T1 (XS40: 6)</b>     | Tool detection sensor signal A             |
| <b>T2 (XS40: 5)</b>     | Tool detection sensor signal B             |
| <b>T3 (XS40: 4)</b>     | Tool detection sensor signal C             |
| <b>T4 (XS40: 3)</b>     | Tool detection sensor signal D             |
| <b>T8 (XS40: 19)</b>    | Tool pan stop and clamp detection signal E |
| <b>TLOCK (XS42: 16)</b> | Tool post push-off signal                  |
| <b>+24V (XS42: 25)</b>  | +24V                                       |
| <b>AGND (XS42: 24)</b>  | 0V                                         |

## Appendix VI Factory Parameters

GSK980TB3/TB3-V factory parameters:

| Parameter No. | A-disk (servo) | B-disk (step) | C and D disks (user -defined) |
|---------------|----------------|---------------|-------------------------------|
| 1             | 00000000       | 00000000      |                               |
| 2             | 00000011       | 00000011      |                               |
| 3             | 00000100       | 00000100      |                               |
| 4             | 00110000       | 00110000      |                               |
| 5             | 01000011       | 01000011      |                               |
| 6             | 01000000       | 01000000      |                               |
| 7             | 00001000       | 00001000      |                               |
| 8             | 11000011       | 11000011      |                               |
| 9             | 01001100       | 01001100      |                               |
| 10            | 00000000       | 00000000      |                               |
| 11            | 00000001       | 00000001      |                               |
| 12            | 00000000       | 00000000      |                               |
| 13            | 00000001       | 00000001      |                               |
| 14            | 10001111       | 10001111      |                               |
| 15            | 00000000       | 00000000      |                               |
| 16            | 00000000       | 00000000      |                               |
| 17            | 1              | 1             |                               |
| 18            | 1              | 1             |                               |
| 19            | 1              | 1             |                               |
| 20            | 1              | 1             |                               |
| 21            | 4000           | 3000          |                               |
| 22            | 8000           | 6000          |                               |
| 23            | 60             | 200           |                               |
| 24            | 60             | 200           |                               |
| 25            | 60             | 60            |                               |
| 26            | 60             | 60            |                               |
| 27            | 8000           | 8000          |                               |
| 28            | 100            | 100           |                               |
| 29            | 60             | 200           |                               |
| 30            | 60             | 200           |                               |
| 31            | 60             | 200           |                               |
| 32            | 100            | 100           |                               |
| 33            | 80             | 80            |                               |
| 34            | 40             | 40            |                               |
| 35            | 40             | 40            |                               |
| 36            | 0              | 0             |                               |
| 37            | 0              | 0             |                               |
| 38            | 1              | 1             |                               |
| 39            | 0              | 0             |                               |
| 40            | 0              | 0             |                               |
| 41            | 1000           | 1000          |                               |

## Appendix VI Factory parameters

|    |          |          |  |
|----|----------|----------|--|
| 42 | 1000     | 1000     |  |
| 43 | 200      | 200      |  |
| 44 | 0        | 0        |  |
| 45 | 0        | 0        |  |
| 46 | 9999999  | 9999999  |  |
| 47 | -9999999 | -9999999 |  |
| 48 | 9999999  | 9999999  |  |
| 49 | -9999999 | -9999999 |  |
| 50 | 10       | 10       |  |
| 51 | 999      | 999      |  |
| 52 | 1024     | 1024     |  |
| 53 | 1        | 1        |  |
| 54 | 1        | 1        |  |
| 55 | 9999     | 9999     |  |
| 56 | 9999     | 9999     |  |
| 57 | 9999     | 9999     |  |
| 58 | 9999     | 9999     |  |
| 59 | 95       | 95       |  |
| 60 | 9999     | 9999     |  |
| 61 | 0        | 0        |  |
| 62 | 1        | 20       |  |
| 63 | 50       | 50       |  |
| 64 | 50       | 50       |  |
| 65 | 500      | 500      |  |
| 66 | 0        | 0        |  |
| 67 | 100      | 100      |  |
| 68 | 10       | 10       |  |
| 69 | 60       | 200      |  |
| 70 | 60       | 200      |  |
| 71 | 0        | 0        |  |
| 72 | 0        | 0        |  |
| 73 | 0        | 0        |  |
| 74 | 0        | 0        |  |
| 75 | 0        | 0        |  |
| 76 | 0        | 0        |  |
| 77 | 0        | 0        |  |
| 78 | 0        | 0        |  |
| 79 | 0        | 0        |  |
| 80 | 0        | 0        |  |
| 81 | 4        | 4        |  |
| 82 | 30       | 30       |  |
| 83 | 1000     | 1000     |  |
| 84 | 15000    | 15000    |  |
| 85 | 200      | 200      |  |
| 86 | 1        | 1        |  |
| 87 | 1        | 1        |  |
| 88 | 500      | 500      |  |
| 89 | 100      | 100      |  |

|     |      |      |  |
|-----|------|------|--|
| 90  | 100  | 100  |  |
| 91  | 100  | 100  |  |
| 92  | 100  | 100  |  |
| 93  | 100  | 100  |  |
| 94  | 100  | 100  |  |
| 95  | 9600 | 9600 |  |
| 96  | 200  | 600  |  |
| 97  | 2    | 2    |  |
| 98  | 2000 | 2000 |  |
| 99  | 0    | 0    |  |
| 100 | 0    | 0    |  |
| 101 | 12   | -45  |  |
| 102 | 30   | 30   |  |
| 103 | 40   | 40   |  |
| 104 | 45   | 120  |  |
| 105 | 0    | 0    |  |
| 106 | 0    | 0    |  |
| 107 | 1000 | 1000 |  |
| 108 | 1000 | 1000 |  |
| 109 | 0    | 0    |  |
| 110 | 0    | 0    |  |
| 111 | 0    | 0    |  |
| 112 | 0    | 0    |  |
| 113 | 0    | 0    |  |
| 114 | 0    | 0    |  |
| 115 | 0    | 0    |  |
| 116 | 0    | 0    |  |
| 117 | 0    | 0    |  |
| 118 | 0    | 0    |  |
| 119 | 0    | 0    |  |
| 120 | 0    | 0    |  |
| 121 | 0    | 0    |  |
| 122 | 0    | 0    |  |
| 123 | 0    | 0    |  |
| 124 | 0    | 0    |  |
| 125 | 0    | 0    |  |
| 126 | 0    | 0    |  |
| 127 | 0    | 0    |  |
| 128 | 0    | 0    |  |
| 129 | 0    | 0    |  |
| 130 | 0    | 0    |  |
| 131 | 0    | 0    |  |
| 132 | 0    | 0    |  |
| 133 | 0    | 0    |  |
| 134 | 0    | 0    |  |
| 135 | 0    | 0    |  |
| 136 | 0    | 0    |  |
| 137 | 0    | 0    |  |

## Appendix VI Factory parameters

|     |      |      |  |
|-----|------|------|--|
| 138 | 0    | 0    |  |
| 139 | 0    | 0    |  |
| 140 | 0    | 0    |  |
| 141 | 0    | 0    |  |
| 142 | 0    | 0    |  |
| 143 | 0    | 0    |  |
| 144 | 0    | 0    |  |
| 145 | 0    | 0    |  |
| 146 | 151  | 151  |  |
| 147 | 0    | 0    |  |
| 148 | 0    | 0    |  |
| 149 | 0    | 0    |  |
| 150 | 0    | 0    |  |
| 151 | 2    | 2    |  |
| 152 | 0    | 0    |  |
| 153 | 0    | 0    |  |
| 154 | 0    | 0    |  |
| 155 | 0    | 0    |  |
| 156 | 0    | 0    |  |
| 157 | 40   | 200  |  |
| 158 | 7600 | 6000 |  |
| 159 | 0    | 0    |  |
| 160 | 0    | 0    |  |