Swiss type CNC lathe

HS-820

operating manual



Guangdong Zhongcong Intelligent Equipment Co., LTD

catalogue

Preface 1
Introduction 6
Note： 6
Contact information： 7
Chapter Ⅰ Safety Matters 7
1.1 General Precautions 7
1.2 Safety Device 8
1.2.1 Mechanical door safety switch 9
1.2.2 Liquid Level detection of lubricating device 9
1.2.3 Cutting Tool Damage Detection 9
1.3 Emergency Stop Button 10
1.4 Safety Device 10
1.4.1 Main Circuit Breaker - Standard 11
1.4.2 Cutting liquid level detection -- Standard 11
1.4.3 Lubricating oil Liquid level detection device -- Standard 11
1.4.4 Air Pressure Protection -- Standard 11
1.4.5 Cylinder in-place detection device -- Standard 12
1.4.6 Interference Check Function - Standard 12
1.4.7 Back shaft jacking test device -- Standard 12
1.5 Matters needing attention during operation 13
1.5.1 Precautions for mechanical Settings 14
Chapter Ⅱ Fundamentals of Machine Tool processing 14
2.1 Overview of Machine Tool 14
2.2 Introduction to the Lathe 15
2.3 Guide 15
2.4 Tool 16
2.4.1 Tool Type 16
2.4.2 Tool Material 19
2.5 Turning and Milling 20
2.6 Numerical control and program 20
2.7 System 20
Chapter Ⅲ Machine Tool Operation 21
3.1 Machine tool Summary 21
3.1.1 Machine Tool Appearance 21
3.1.2 Machine tool structure 22
3.1.3 Machine tool size (HS320/HS820/HS926 universal) 20
3.1.4 Machine tool parameters 21
3.2 Machine tool switch 23
3.2.1 Main Power switch 23
3.2.2 System Power Switch 23
3.2.3 Emergency Stop Button 23
3.3 Boot Operation 24
3.4 Shutdown Operations 24

3.5 Emergency Stop 25
3.6 Operation Panel 25
3.6.1 A Displaying Part 26
3.6.2 C Functions with number button 30
3.6.3 Select Mode 30
3.6.4 Multiplier Switch 32
Chapter Ⅳ Function and operation of CNC 36
4.1 Screen switch 34
4.1.1 Screen 34
4.1.2 Main course menu key 35
4.2 Processing information Settings 40
4.2.1 Machine Monitoring area 40
4.2.2 Program Code monitoring area 38
4.2.3 Processing information display area 38
4.3 Processing Record Form 40
4.4 Clear processing time 42
4.5 MDI Enter 43
4.6 Program Editing 45
4.6.1 Loading program edit 45
4.6.2 (G,M,T) code screen list 46
4.6.3 Deleting Line 48
4.6.4 Search for/Replace 49
4.6.5 Graphics assisted input 51
4.7 File Management 60
4.7.1 Opening a New File 60
4.7.2 Copying a File 62
4.7.3 Deleting File 64
4.8 File Transfer 65
4.8.1 File Input 65
4.8.2 File Output 69
4.9 Frame coordinates 70
4.10 Tool Management 71
4.10.1 Wear input 71
4.10.2 Shape input 73
4.10.3 Setting the Knife Tip 75
4.10.4 Tool length input 76
4.10.5 Tool Life Management 77
4.11 Maintaining 80
4.11.1 Alarm display 80
4.11.2 System Service life 82
4.12 User Parameters 84
4.13 Function Settings 85
4.13.1 Switching On/Off with or Without guide bushing 85
4.13.2 Clip Change Mode Switch On/Off 87
4.13.3 Tool life On/Off 88
4.13.4 Use On/Off for feeder 89

4.13.5 Standby On/Off 100
4.13.6 Run empty On/Off 91
4.13.7 Power Head Maintenance timing 92
4.13.8 Auxiliary Panel 94
Chapter Ⅴ Tool installation 105
5.1 Tool row Installation 97
5.2 Cutter installation of milling unit 100
5.3 Extension of the end milling unit 104
5.4 Installing a Fixed Drilling Tool 104
5.5 Switch the Fixed Drilling Tool 108
5.6 Installing a Rear Tool 108
5.6.1 Install the "Measurement X2" axis of T21 turning tool 109
5.6.2 Install "Measuring Z2" shaft of T21 turning tool 111
5.6.3 Installing the "Measure Z2" shaft of the T22 twist drill 112
Chapter Ⅵ Programming instructions 126
6.1 NC Program 115
6.2 Programming of numerical control programs 116
6.2.1 Check Drawings 116
6.2.2 Tool Configuration 116
6.2.3 Processing Configuration 117
6.2.4 Coordinate value 117
6.2.5 Programming 117
6.2.6 Processing data 117
6.3 Program Structure 117
6.4 Coordinate system and travel of each axis 119
6.5 Function Instructions (G code) 122
6.6 Auxiliary Instructions (M Code) 124
6.7 Tool instruction (T code) 12638
Chapter Ⅶ Description of G code 140
7.1 Quick Locating (G00) 141
7.2 Straight Line Interpolation (G01) 127
7.3 Circular Interpolation (G02 G03) 129
7.4 Pause (G04) 131
7.5 Synchronization Wait (G04.1) 132
7.6 Cylindrical Interpolation (G07.1) 133
7.7 Do Stop Testing (G09) 135
7.8 Programming Data Input (G10) 135
7.9 Polar Coordinate Interpolation On/Off (G12.1 G13.1) 136
7.10 Plane Selection (G17,G18,G19) 139
7.11 Setting of British and Metric Units (G20 G21) 140
7.12 Automatic Reference Point Retrieval (G28) 141
7.13 Reference Point 2 Restoration (G30) 141
7.14 Thread Cutting (G32) 142
7.15 Cutting of Variable Pitch Thread (G34) 146
7.16 Tool Tip R correction (G40,G41,G42) 148

7.16.1 Z-X Plane rectification Method 148
7.16.2 Correcting Login Data 149
7.16.3 Starting a Program segment 149
7.16.4 Canceling a Program Segment 150
7.16.5 Program Examples 151
7.16.6 Program Examples 151
7.17 Coordinate System Setting and Spindle Locking Speed Setting (G50) 152
7.17.1 Setting the coordinate system 152
7.17.2 Setting the spindle Locking Speed 152
7.18 Polygon Cutting Start/Cancel (G50.2 ~ G51.2) 153
7.18.1 Turn on polygon cutting 153
7.18.2 Turn off polygon cutting 153
7.19 Bringing Up Macro Programs (G65 ~ G67) 153
7.19.1 Simple Program Import 153
7.19.2 Importing macro Commands 155
7.19.3 Simple Switch Out (G65) 155
7.19.4 Mode Call Out (G66,G67) 155
7.19.5 Specifying Variables 156
7.19.6 Variable specified Ⅰ 156
7.19.7 Variable specified Ⅱ 156
7.19.8 Macro Usage Instructions 157
7.19.9 Macros Classification 157
7.19.10 Macros Classification 158
7.19.11 Public Variables 158
7.19.12 Area Variables 158
7.19.13 Operator 159
7.19.14 Syntax Description 159
7.19.15 Monitoring Area Variable Addresses 164
7.19.16 Argument description 165
7.19.17 Function Table 165
7.20 Compound Fixed Cycle (G70 ~ G76) 169
7.20.1 Fine Cycle (G70) 170
7.20.2 Lateral (Outside Diameter) coarse cycle (G71) 172
7.20.3 Radial (end face) coarse cycle (G72) 175
7.20.4 Forming contour rough turning cycle (G73) 177
7.20.5 End Cutting Cycle (G74) 180
7.20.6 Lateral (X-axis) peck cutting cycle (G75) 181
7.20.7 Compound Thread Cutting Cycle (G76) 185
7.21 Drilling fixed circulation (G80,G83 ~ G87,G88) 187
7.21.1 Canceling the Drilling Fixed Circulation (G80) 187
7.21.2 Front Drilling Cycle (G83) 188
7.21.3 Positive Forward Tapping Cycle (G84) 1909
7.21.4 Side Drilling Circulation (G87) 194
7.21.5 Forward Side Tapping Cycle (G88) 195
7.22 Single Fixed Cycle (G90,G92,G94) 198
7.22.1 Inside and Outside Diameter Turning Cycle (G90) 198

7.22.2 Thread Cutting Cycle (G92) 201
7.22.3 End Cutting Cycle (G94) 204
7.23 Cycle speed Control (G96,G97) 207
7.23.1 Cycle speed must be controlled ON command 207
7.23.2 Cycle speed must control OFF instruction 207
7.23.3 Setting the maximum rotational speed of shaft 207
7.24 Feed function specification (G98,G99) 208
7.24.1 Each Minute Feed (G98) 2089
7.24.2 Each Turn Feed (G99) 208
7.25 Main Axis and Phase synchronizations (G114.1), cancel synchronizations (G113) 208
7.25.1 The Rotational speed is the same 208
7.25.2 Phase Synchronization 210
7.26 C1,C2 concurrent period (M1000), Cancel concurrent period (M1001) 213
7.27 Z1,Z2 concurrent (M2000), Cancellation concurrent (M2001) 216
7.28 Setting Coordinate System (G150) 2191
7.29 Processing Intelligence Settings (G299) 221
7.30 Automatic Cut-off (G300) 221
Chapter Ⅷ Programming examples 244
8.1 Processing Configuration 222
8.2 Preparation for Processing 225
8.3 Center hole processing 225
8.4 Drilling 226
8.5 Tapping processing 2270
8.6 Outside diameter of coarse vehicle 227
8.7 Lathe tooth processing 228
8.8 Outside diameter of finishing car 229
8.9 Groove processing 230
8.10 Milling 231
8.11 Machining side center hole 232
8.12 Side drilling 233
8.13 Machining of rear outer diameter 235
8.14 Cut off and connect the auxiliary shaft 236
8.15 Machining the outer diameter of the back 237
8.16 Drilling holes on the rear side 238
8.17 Machining the inner hole on the back 238
8.18 Rear Grooving 239
8.19 Discharge materials 240
Chapter Ⅸ Maintenance 267
9.1 Routine Maintenance 241
9.2 Regular Maintenance 241
9.3 List of machine consumables 242
Chapter Ⅹ Troubleshooting 269
10.1 Contents and handling of system alarm 243
Attached Table I MLC alarm cause and disposal 274

Drive Alert -ALARM 274

Introduction

Thank you for buying Zhongcong Technology HS-926/ HS-820 centering type precision CNC lathe.

This book is the operation manual of HS-926/ HS-820 centering precision CNC lathes, which explains the operation, commissioning and maintenance of the machine in detail. In order to prevent accidents or failures, please read and fully understand the detailed contents of this book and other attached instructions before operation.

The machine is driven by dangerous voltage and contains multiple moving parts. Failure to operate in the correct way or follow the safety precautions will result in the loss of the machine or other property.

This book cannot fully cover all the actual conditions related to operations. Please interpret the permitted operations listed in this book as prohibited operations.

Note

1. Although the contents of this book are perfect, there may still be mistakes caused by omissions. If found, please contact our company in time.
2. The contents of this book are subject to change without prior notice due to the continuous improvement of the machine.
3. The contents of this book may be inconsistent with the actual machinery.
4. Guangdong Zhongcong Intelligent Equipment Co., Ltd. owns the copyright of this book. Without the prior written consent of the Company, any reproduction or transmission of any content of this Book by any means is prohibited.
5. The company and product names recorded in this book are trademarks or registered trademarks of each company.

Contact information

If you have any questions about this machine or this book, please contact us.

Guangdong Zhongcong Intelligent Equipment Co., LTD

Phone number：0769-39029879

Safety mark

Please read this manual carefully or all kinds of safety labels pasted on the machine. On the basis of full understanding, start operation preparation, operation and maintenance work. Security identifier warning types are as follows:

Hazard: Indicates a defined hazardous condition which, if not avoided, will result in death or serious injury.

Warning: This is a potentially hazardous situation which, if not avoided, could result in death or serious injury.

Caution: If not avoided, it may result in injury, mechanical damage and other conditions.

Chapter Ⅰ Safety Matters

Before running the machine, please observe the following precautions to ensure the safety of the operator.

1.1 General Precautions

⚠WARNING

- When operating machinery, please wear uniform and safety shoes correctly.
- maintain the mechanical and the surrounding clean, alert for an accident, nip in the bud.
- prohibited remove the cover and safety devices.
- protective cover and the cover plate is an important part of security. During maintenance, be sure to turn off the external power supply (transformer). Before turning on the power again, ensure that the cover plate and protective cover are correctly installed.
- Modification of machinery may cause mechanical failure or endanger operator safety. If modification is necessary under special circumstances, please follow our company's instructions to ensure safety.
- It is forbidden to open the mechanical protective door in the process of mechanical operation. Automatic operation is prohibited when the protective door is opened.
- When running automatically, please be sure to monitor the running state (chip, cutting oil, tool, etc.) at appropriate intervals and deal with improper conditions in time.
- When adjusting, measuring or cleaning, do not touch the tool or workpiece in operation with your hands. Please be sure to do it in the state of shutdown.
- Maintain sufficient cutting oil for cooling during operation.
- Pay attention to safety precautions when installing optional components.
- Never run machinery with its parts removed.

⚠PAY ATTENTION

- Please be sure to maintain and clean all parts of the machine regularly, pay attention to mechanical damage and loose screws and deal with them in time.
- The machinery must be properly grounded.
- Before starting the machine, check the tool, fixture, chuck, bushing, etc., to ensure that they are in the correct position.

1.2 Safety device

1.2.1 Mechanical door safety switch

When the safety door test is opened, the system detects that the spindle guard or cutting chamber door cannot operate automatically when it is opened. The "open door" button can be used to disable the safety door switch. Please be sure to operate it manually when it is safe.

⚠ WARNING

- Operation automatically, please do not disable the safety door switch test.
- When using the "open door" button to disable the safety door switch, please make sure that it is safe before running.

1.2.2 Liquid level detection of lubricating device

The lubricating oil unit is equipped with a lubricating oil level detection device.

When the oil in the lubricating oil box is low, the lubricating oil level detection device will act, the system screen will give corresponding prompts, and the machine will enter the single-cycle stop state.

1.2.3 Cutting Tool Damage Detection

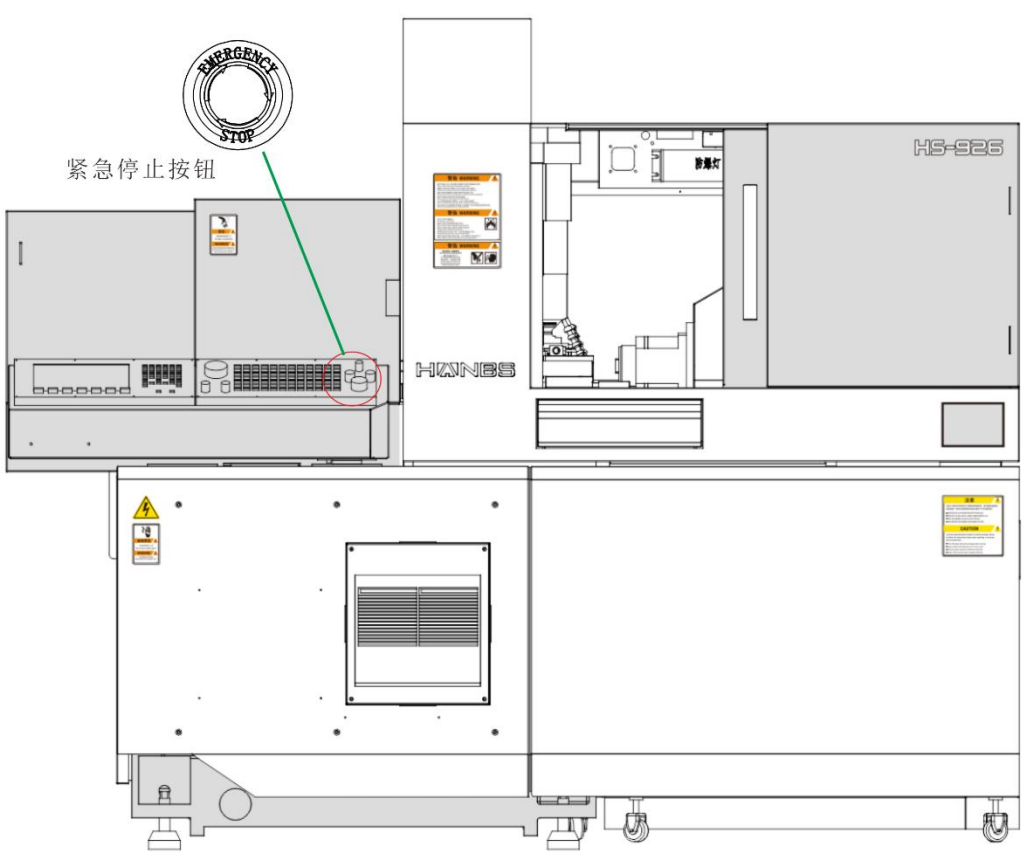
This device is a device for detecting cutter damage by confirming whether there is product residue near the guide sleeve at the end of the cutting process. When the cutting knife damage is detected, the alarm will be displayed on the screen.

⚠ WARNING

- Cutting knife damage will lead to mechanical damage or fire. Please replace it immediately after it is damaged.
- When processing new products, please be sure to confirm the operation of this device.

1.3 Emergency stop button

There is a red emergency stop button on the operation panel. Press this button to stop the operation of the machine before leaving the machine in case of emergency such as fire, earthquake or lightning strike, and other emergency evacuation situations. In addition, in the process of operation, if you perceive any dangerous situation, please also press this button. But it is worth noting that if the machine is running, pressing the emergency stop button will not only damage the product being processed, but also may damage the tool itself. Before resuming the emergency stop state, it is necessary to confirm whether it is safe. Then, press the emergency stop button in the clockwise direction to remove the emergency stop state, turn off the power of the machine once, and then switch it on immediately. In addition, please use the hand wheel mode to restore all the movable parts of the machine to the backward point, and take out all the materials being processed in the machine.



1.4 Safety device



警告

对于各种事故和危险状况，安全装置并不能确保绝对地安全，使用本机时，请务必遵守本章中的安全注意事项。如果不遵守安全注意事项，有可能导致死亡事故或重伤事故。

In order to prevent all kinds of accidents that may occur during use and to find potential dangerous conditions, the machine is equipped with safety devices, standard safety devices and various safety devices are as follows:

- Check the devices that may cause accidents during machine operation.
- A device that stops the machine when dangerous conditions occur.
- Devices to prevent the appearance of defective products.
- Devices that prevent damage to machinery or cutting tools.

Safety devices are very effective for the following situations;

- When the operator's attention may be distracted during night or continuous operations.
- When strengthening accident prevention measures.

Comment:

Safety devices for optional items can only be used after the device is purchased.

1.4.1 Main Circuit Breaker - Standard

If the current is detected more than 30A or leakage more than 30mA, the machine will stop automatically.

1.4.2 Cutting liquid level detection -- Standard

Installed on the cutting tank, detection of cutting oil oil is sufficient device, if the liquid level is lower than the lower limit, the system will alarm, in order to prevent fire and mechanical damage accidents, processing 1 cycle after the end of the machine will automatically stop.

1.4.3 Lubricating oil Liquid level detection device -- Standard

If the liquid level is lower than the prescribed lower limit, the system will alarm. In order to prevent mechanical damage accidents, the machine will automatically stop after 1

cycle of processing.

1.4.4 Air pressure protection - Standard

In order to protect the bearing of the spindle and guide sleeve device as well as the collision, the pressure switch device is used to detect whether the air pressure is normal. If the air pressure is too high or too low, the system will alarm and the machine will automatically stop immediately.

1.4.5 Cylinder in place detection device - Standard

The device installed on the cylinder block detects whether the cylinder is running in place. If the cylinder execution is abnormal, the system will give an alarm and the machine will automatically stop immediately.

1.4.6 Interference check function - Standard

Check whether the machine parts and workpiece between the function of interference, in the process of machine operation, if the possibility of interference is checked, the system will appear alarm, in order to prevent machine damage accident, the machine will automatically stop immediately. If these alarms appear, the operator must correct the procedure.

1.4.7 Back shaft jacking test devices -- Standard

Check whether the back shaft spitting device is normal, in the process of machine spitting, if there is abnormal spitting, the system will alarm, in order to prevent the machine damage accident, the machine will automatically stop immediately.

1.5 Matters needing attention during operation

During the operation of the machine, please be sure to observe the following precautions.



危险

有可能造成死亡或重伤事故，请务必遵守以下注意事项。

▲请不要在机器运行时打开机器的前门和后门。
▲在机器运行过程中，请不要触摸和接近切削刀具或正在运动的机械部位，当机器顶部绿灯亮时或系统操作面板启动灯亮时，即表示机器正处于<运转中>的状态。
▲在机器运行过程中，请不要触摸电气系统的工作部位，如果触摸，可能会引起触电的事故。在主电路断路器处于ON状态时，绝对不能打开机器控制面板的罩或电箱门，如果要打开，请务必在断电状态或专业人士的指导下进行操作。
▲绝对不允许对机械和控制电路进行改装。
▲当紧急事态发生时，按下紧急停止按钮，让整个机器处于停止状态，在松开紧急停止按钮前，请首先确认机器是否处于安全状态，然后再松开，排除故障之后在进行下一步操作。



警告

在机器运行过程中，请务必遵守以下注意事项，否则有可能造成死亡或重伤事故。

▲请不要在机器运行时打开所有与安全有关的装置。
▲在机器停止运行之前，请不要打开正面所有的门。
▲请在确认了正面左右所有的门都已经关闭，而且与安全有关的所有装置都打开之后，开始运转机器。
▲操作机器时，请在充分理解操作内容的基础上，用目视确认了需要使用的操作开关后，再按动启动开关。
▲在进行切削加工时，如果所用的材料属于易燃材料，由于这种材料引发火灾的可能性很大，所以操作者应在操作过程中，保持高度警惕，随时进行监控。



注意

有可能造成死亡或重伤事故，请务必遵守以下注意事项。

▲务必将机器正确接地，电源线和接地线有可能成为对机械产生影响的电气干扰源，通过适当的接地可以排除这种干扰，请参照维护说明书。
▲请确认切削油的油量是否充足，必要部位的润滑供应情况是否合乎要求。
▲请在启动机器前，对刀具进行检查，确认刀具是否被正确地安装在指定位置。
▲请务必认真阅读本说明书中记载的定期点检事项。
▲请对切削周围以及切削油箱的渣子进行清理。

1.5.1 Precautions for mechanical Settings .

Please be sure to observe the following precautions. Otherwise, death or serious injury may result.

•When setting up or moving machinery, cranes, forklifts or lifting slings are needed. When using such tool vehicles, the operators should hold the qualification certificates recognized by the State authorities.

•Turn off the main circuit breaker of the machine and the circuit breaker of the power supply at the factory before connecting or removing the power cable to the machine.

•When mechanically connecting power cables, install ground cables.

Chapter Ⅱ Foundation of machine tool processing

2.1 Machine tool overview

Machine tools usually refer to machines that process raw materials into parts. The parts of machine tools are usually processed by machine tools, so machine tools are also called "mother machines". Most of the industrial products we commonly come into contact with are processed using machine tools.

There are various types of machine tools, and usually the ones with more contact can be roughly divided into the following two types:

1. Milling machine. Milling machine is a machine tool for cutting fixed materials with rotary cutting tools. Milling machines that are usually more functional and equipped with numerical control systems are called "machining centers".
2. A lathe is a machine tool that uses a fixed tool to cut a rotating material. Lathes equipped with numerical control systems are often referred to as "CNC lathes" to distinguish them from traditional lathes.

2.2 Introduction to the Lathe

According to the structure of the lathe can usually be divided into the core type and the tool type two. The main difference is the processing way in the direction of the length of the product: the core lathe can be processed by the spindle driven material movement, the tool walking type spindle fixed by the way of moving the tool.

There are three main elements related to machining in lathe: fixture, material and tool. This machine belongs to the small lathe, usually use the chuck as a fixture to clamp the material, and drive the material to rotate.

If the machining part is far away from the clamping position, it will be difficult to ensure the machining accuracy due to the bending or machining force of the material. Therefore, usually processing, material elongation and material diameter ratio in 2.5-3 is appropriate.

For the processing of slender products, the material support mechanism (guide sleeve) is installed between the chuck and the tool, so that the chuck can be moved in the direction of the length of the material of the machine called Swiss automatic lathe, also known as the core lathe. This machine is such, because the guide sleeve near the tool support such, can reduce the bending caused by the force of the material during processing, so as to achieve high precision processing.

2.3 Guide

Guide sleeves are usually divided into fixed guide sleeves and rotating guide sleeves.

The inner ring of the fixed guide sleeve is fixed and the supporting material rotates. The material and the inner ring of the guide sleeve will simultaneously produce friction in the direction of rotation and longitudinal, and the contact surface is usually coated with oil film to reduce friction. Fixed guide sleeve is usually used for processing watches and watches fine small parts, processing roundness is higher.

The inner ring of the rotating guide sleeve can be rotated with the material. The main friction between the material and the inner ring of the guide is generated in the longitudinal direction. The machine adopts synchronous rotating guide sleeve, the guide sleeve speed is the same as the spindle speed, there is no friction in the rotation direction.

2.4 Tool

2.4.1 Tool type

There are a variety of tools used in machine tools. Different tools need to be selected according to the actual processing type. Commonly used tools are as follows:

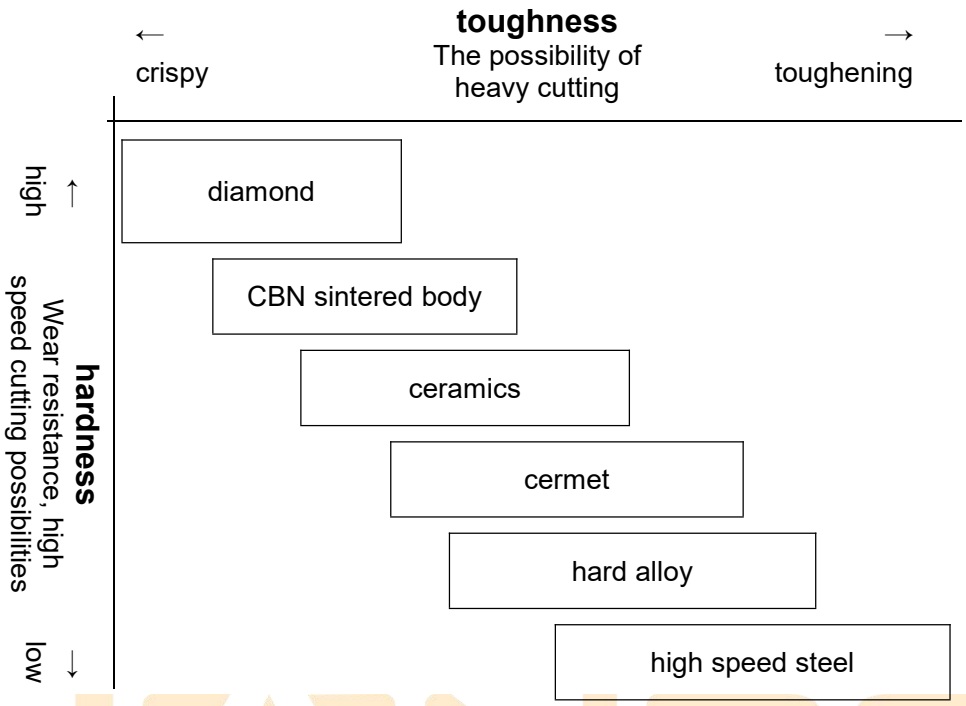
- before sweeping knife: about the way of products from the front to the back shape of cutting tool.
- after sweeping knife: about the way of products from the back side shape of cutting tool.
- slot knife: for cutting tool products groove part.
- cutting knife: used for products from raw materials to cut off the tool, appearance similar to slot cutter, single point more prominent, for further processing.
- center drill: for products center hole processing tool, usually before drilling machining center hole so that the drilling process is more stable.
- Bit: drilling tool products.

2.4.2 Tool material

The materials of the tools used in machine tools also vary. In order to ensure the product processing effect, it is necessary to choose the appropriate tool according to the different materials of the product. Usually the material of the tool has the following categories:

- Diamond, a tool for pressing and forming single diamond crystals.
- CBN sintered body is a polycrystalline sintered body sintered by adding cobalt, titanium carbide and other materials on cubic boron nitride.
- Ceramic, alumina, titanium carbide, hydrogen nitride and other sintered ceramic tool.
- Cermet, sintered alloy made by powder metallurgy with a mixture of nickel added to titanium carbide and titanium nitride.
- Hard alloy, sintered alloy made by powder metallurgy technology after adding titanium carbide, titanium nitride and cobalt to tungsten carbide.
- High speed steel, alloy tool steel with iron as the base material and tungsten, chromium, vanadium, molybdenum, etc.

Characteristics of main tool materials



2.5 Turning and milling

Lathe processing is mainly the use of cutting tools and rotating material contact, this kind of processing is called turning. The corresponding method of machining materials with a rotating tool is called milling.

Milling on a conventional lathe is usually done through a secondary machining process because it is usually not possible to use rotary tools for machining. This kind of lathe like this machine integrated milling unit can be directly on the machine to achieve milling processing.

2.6 Numerical control and program

Numerical Control (NC, Numerical Control) is an abbreviation of digital control, which means that the machine tool can be controlled by preprogrammed action instructions. This collection of action instructions for a machine tool is called a program.

2.7 system

The system referred to in a machine tool usually means the operating system that controls the operation of the machine tool. For the convenience of programming and user understanding, usually only a single system is used to control a set of axes and related

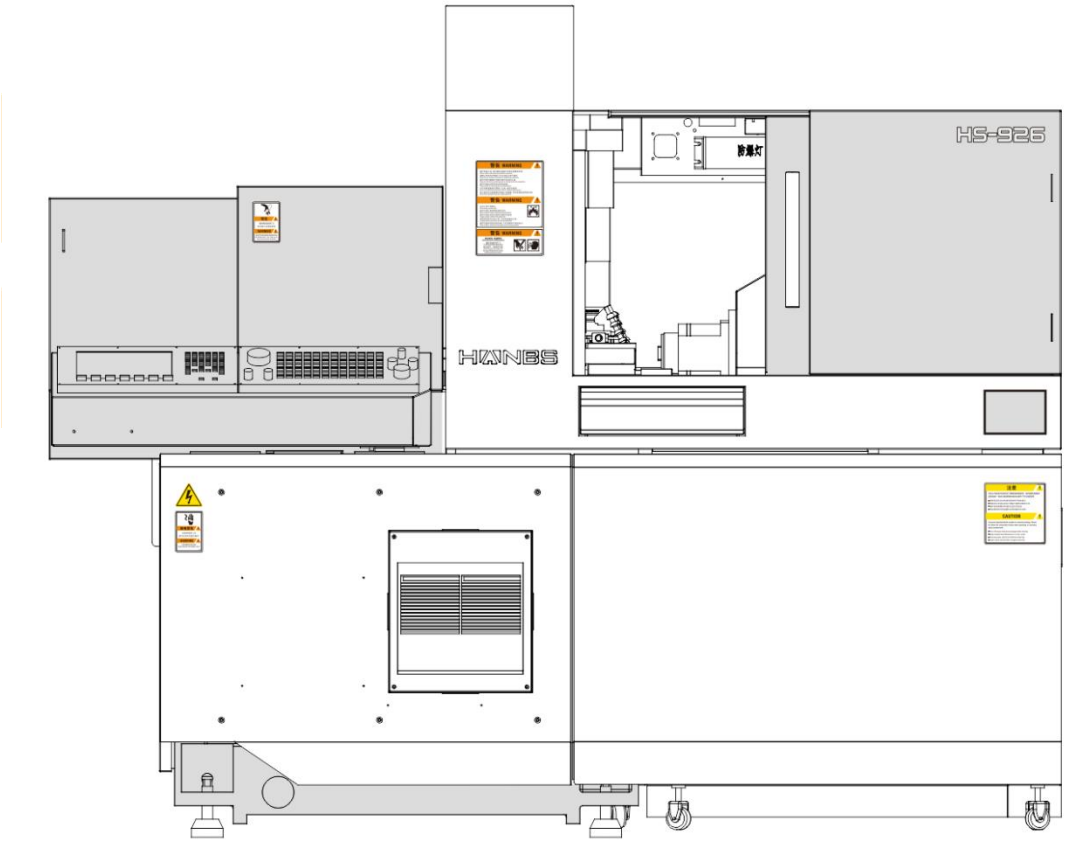
functions. Due to the large number of axes, there are multiple groups of axes, so the form of double system is adopted.

In a dual system, $1 represents system 1 and $2 represents system 2. Each system usually executes the program of each system independently, and can also make the two systems cooperate and coordinate through the system association instruction.

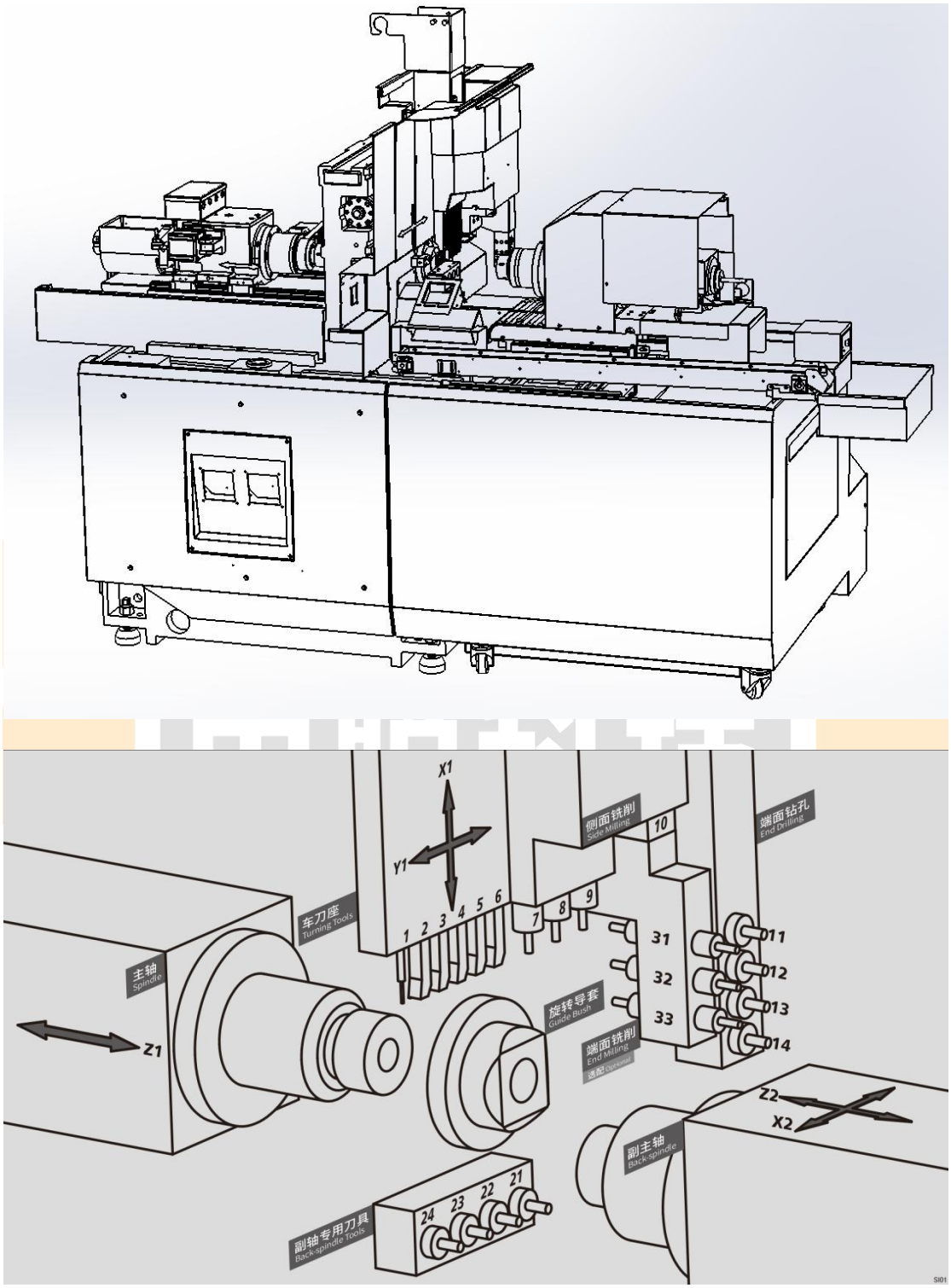
Chapter Ⅲ Machine tool operation

3.1 Machine tool summary

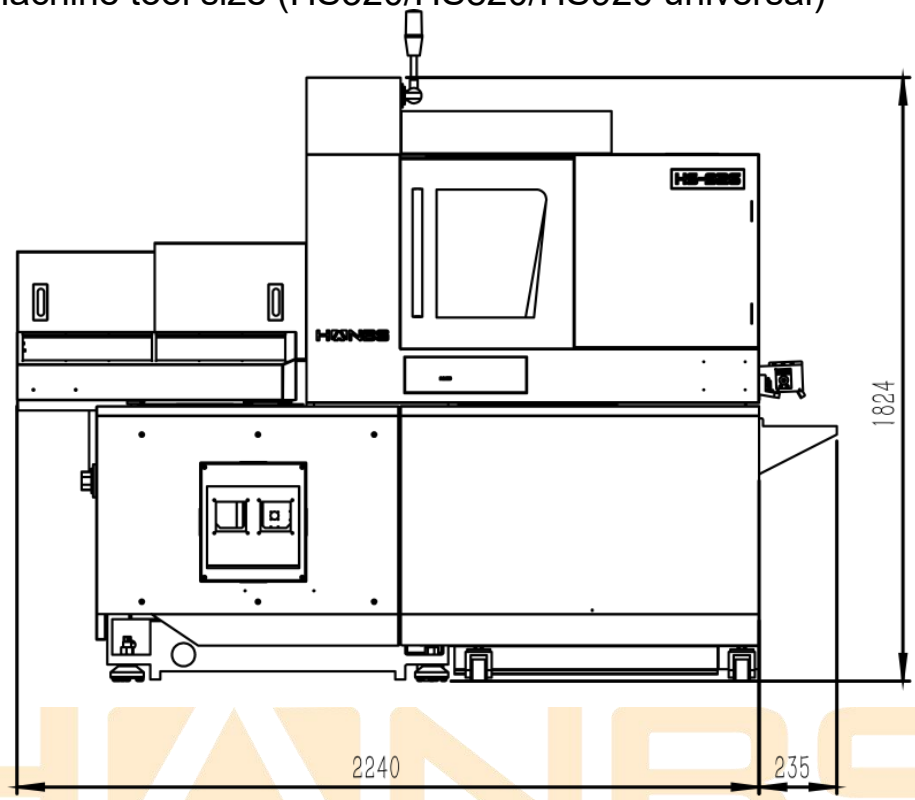
3.1.1 Machine tool appearance

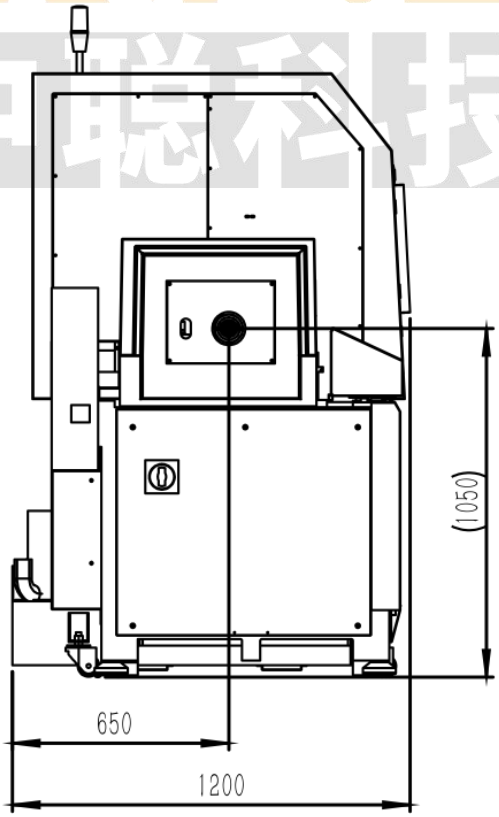


3.1.2 Machine tool structure



3.1.3 Machine tool size (HS320/HS820/HS926 universal)





3.1.4 Machine tool parameter

HS926

Maximum machining diameter（D）	Ø26mm
Maximum machining length (2.5D without bush mode)	190mm
Maximum axial bore diameter	Ø10mm
Maximum axial tapping size	M8
Maximum spindle speed (for short time only)	10000rpm
Radial maximum bore diameter	Ø7
Radial maximum tapping size	M6
Maximum clamping diameter of the back spindle	Ø26mm Back spindle maximum workpiece extension:50mm
Maximum borehole diameter of back spindle	Ø8mm
Maximum tapping diameter of back spindle	M6
Maximum spindle speed on back side (for short time only)	12000rpm
Maximum workpiece removal length	100mm
Side power tool unit	4
Row cutter（□12×120×5+□16×120×1）	6
Fixed tools (shared front and back)	Ø25mm×4×2；backØ25mm×4 12
Standard maximum number of tool installations	22
Z1, Z2, X2, Y-axis feed velocity	30m/min
X1 axis feed velocity	24m/min
Spindle center height	1060mm
Input power	10.0kW
weight	2600kg
size（W×D×H）	2250×1200×2000mm

HS820

Maximum machining diameter（D）	Ø20mm
Maximum machining length (2.5D without bush mode)	190mm
Maximum axial bore diameter	Ø10mm
Maximum axial tapping size	M8
Maximum spindle speed (for short time only)	10000rpm
Radial maximum bore diameter	Ø7
Radial maximum tapping size	M6
Maximum clamping diameter of the back spindle	Ø20mm 背面主轴最大工件伸出 50mm
Maximum borehole diameter of back spindle	Ø8mm
Maximum tapping diameter of back spindle	M6
Maximum spindle speed on back side (for short time only)	12000rpm
Maximum workpiece removal length	100mm
Side power tool unit	4
Row cutter（□12×120×5+□16×120×1）	6
Fixed tools (shared front and back)	Ø25mm×4×2；背面Ø25mm×4　12
Standard maximum number of tool installations	22
Z1, Z2, X2, Y-axis feed velocity	30m/min
X1 axis feed velocity	24m/min
Spindle center height	1060mm
Input power	9.60kW
weight	2500kg
size（W×D×H）	2250×1200×2000mm

3.2 Machine tool switch

3.2.1 Main power switch

state	description
I-ON	Power on the machine
O-OFF	Power off the machine



The main power switch is the power switch of the machine.

3.2.2 System power switch

System power switch Power on the control system.

Name	description
Power on	Power on the control system
Power off	Cut off the power supply to the control system

3.2.3 Emergency stop button

In case of emergency, press this button to stop the machine immediately. This button can also be used to cut off the power supply of the machine when it is necessary to end the operation of the machine.

3.3 Boot operation

Before processing, please start the machine in the following order

1. Remove all parts of the machine and chip box accumulated chips.
2. Check whether there are foreign bodies inside the machine to prevent collision.
3. Check whether the tool is in a safe position.
4. Check whether cutting oil, lubricating oil and spindle cooling oil are sufficient.
5. Open the intake valve located on the back of the fuselage.
6. Turn ON the main switch and rotate it to the ON position.
7. Press the power button on the operation panel and watch the screen until the system is started.
8. Turn the EMERGENCY STOP button to the right to cancel the emergency stop.

⚠ PAY ATTENTION

- If any exception is found during startup, please contact technical personnel.
- After starting the machine, ensure that the tool is in a safe position.
- Before starting the machine, check the tool, fixture, chuck, bushing, etc., to ensure that they are in the correct position.

3.4 Shutdown operation

When the machine is running, please shut down the machine in the following order;

1. Ensure that the machine tool is in single operation mode.
2. Wait until the program is finished (the processing is finished and the program head is returned). The machine tool alarm light is yellow.
3. Press the EMERGENCY STOP button.
4. Press the power button on the operation panel. The screen disappears.
5. Turn OFF the main switch and turn it to the OFF position.
6. Close the intake valve on the back of the fuselage.。

⚠ PAY ATTENTION

- Please follow the normal operation procedure before shutdown.
- In case of shutdown under special circumstances, please inform the technician first.
- When the main power supply is turned off in the rotation of the main and auxiliary shafts, the main and auxiliary shafts will continue to rotate due to inertia. Therefore, when the main power supply is turned off directly in the

rotation of the main and auxiliary shafts, and the door of the spindle room or cutting room needs to be opened, please wait more than 1 minute before opening.

3.5 Emergency stop

In case of an EMERGENCY, press the EMERGENCY STOP button immediately. After the emergency is relieved, please pay attention to the following when starting the machine again:

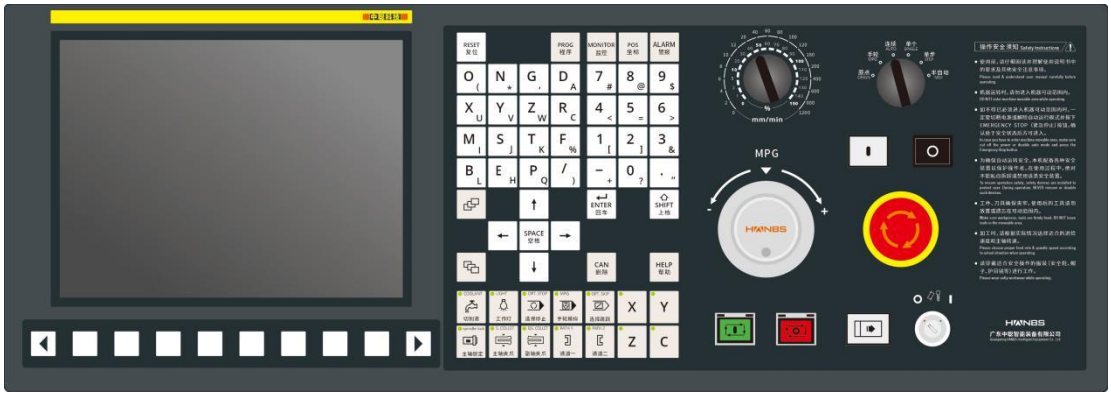
1. Check and rectify the fault.
2. Remove all knives from the material.
3. Check whether the tool is in good condition.
4. Cut off unfinished products.
5. Make sure that the material cutter stays in the material position, open the handwheel simulation button on the operation panel, and execute the program completely in the handwheel simulation mode before the execution. If no abnormal, then carry on continuous automatic processing.

⚠ PAY ATTENTION

- Make sure that the machine is in a safe position before lifting the emergency stop and restarting.
- When the emergency stop is pressed in the rotation of the main and auxiliary shafts, the main and auxiliary shafts will continue to rotate due to inertia. Therefore, when you need to open the spindle room or cutting room door after pressing emergency in the rotation of the main and auxiliary shafts, please wait more than 1 minute before opening it.

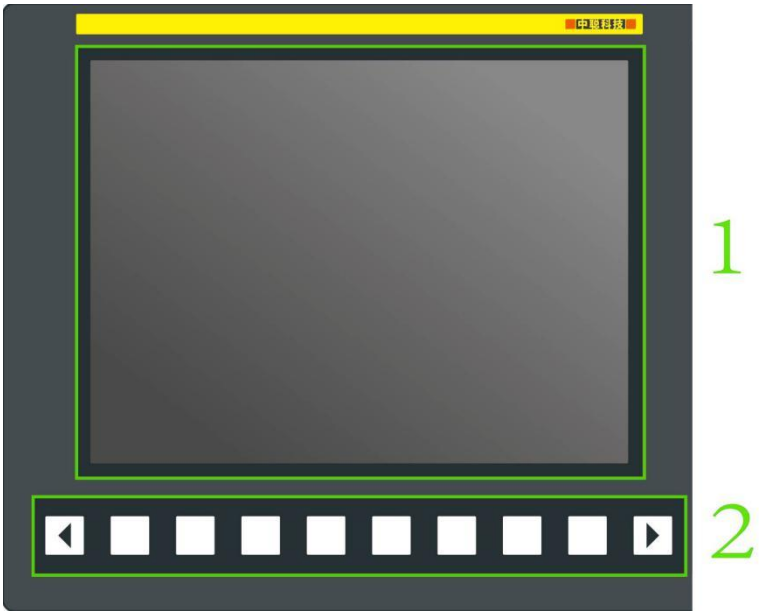
3.6 Operating panel

Description of each function area on the operation panel



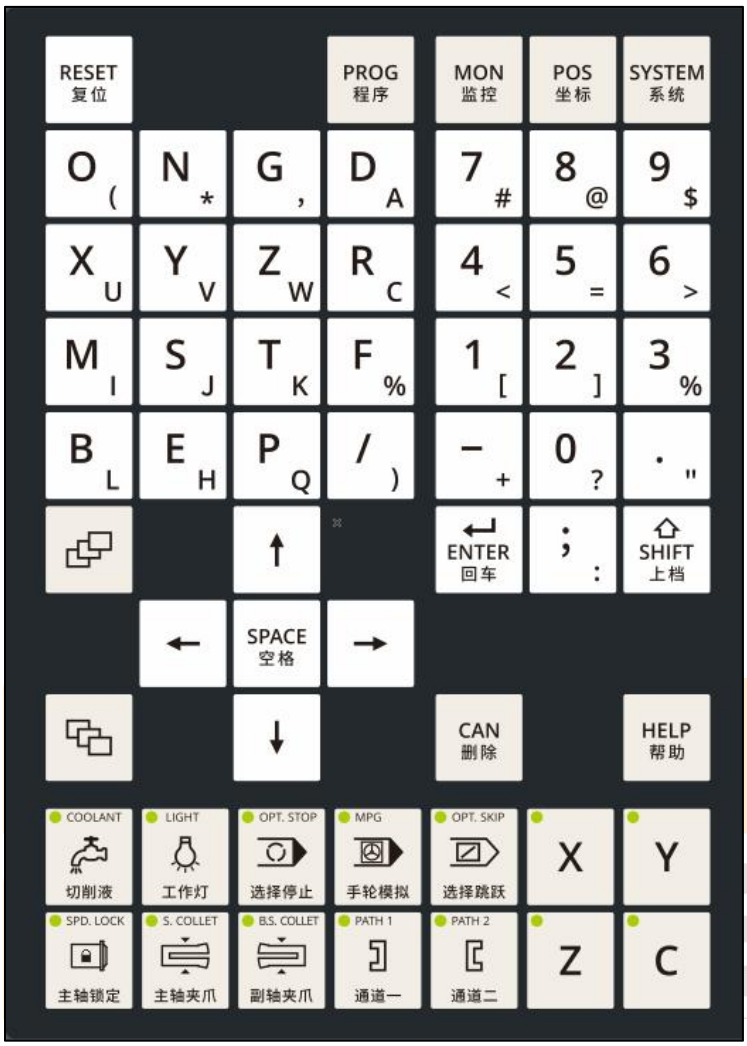
A **B** **C**

3.6.1 A Display section



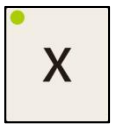
1	Display area	Displays the current system content.
2	Menu button	The leftmost button is used to turn the page back or left, the rightmost button is used to turn the page down, and the middle button corresponds to the function displayed at the bottom of the screen.

3.6.2 C Function with digital button



Description of panel operation function keys

Mark	Name	Function
	Cutting oil	This button can be used to turn on/off the "Cutting oil". Note: Except in the "continuous" and "single" modes, please switch the mode to "single step" if you want to turn on/care about cutting oil during the execution of the program.
	Working lamp	This button can be used to turn on/off the working light.

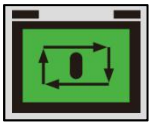
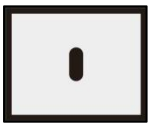
	Select stop	This function can be used to set whether "M01" in NC program is stopped.
	Hand wheel simulation	"Automatic operation" mode and "MDI" mode, after opening the hand wheel simulation, the movement of the hand wheel can be used to control the execution of the program segment.
	Single section execution	This function is for the user to program a "single section" check.
	Spindle locking	This function key is used to enable the C-axis on/off in Manual mode.
	Master and auxiliary/gripper	Main and auxiliary shaft clamp jaws open/close.
		
	Channel 1/ Channel 2	Select the channel to be selected in MDI, Auto Run, and Hand Wheel modes. (Note: Channel 1 and channel 2 are lit at the same time for two-channel processing, and only single channel can be lit when the handwheel moves in the direction).
		
	Directional move key	The machine motion direction control key, "X+,X-,Z+,Z-,Y+,Y-,C+,C-", selects the axis to be moved. Only one axis can be selected at a time. (Note: The auxiliary axis can be moved in channel 2 mode)
		
		
		

标志	名称	功能
RESET 复位	reset	Press the reset key for the CNC internal and perform the following items; ·Cancel move instruction ·Clear the input field (valid in EDIT mode) ·Remove the cause and cancel the alarm ·Deauxiliary function (M function) ·G code reset ·stoppage
PROG 程序	program	The related screen of Program Editing is displayed
MON 监控	Processing monitoring	This key displays processing speed, processing time, manual data entry (MDI), and information commonly used in processing
POS 坐标	coordinate	Display machine coordinates
SYSTEM 系统	System function	Display system content, except "alarm information, parameter information" Etc.
ENTER 回车	Carriage return	Used to wrap lines when entering data
SHIFT 上档	upgear	Press this key before entering the text in the lower right corner of the address/number key
CAN 删除	delete	Deletes a text before the cursor position in the data setting field
HELP 帮助	help	Display operation guidance, parameter guidance, alarm guidance and so on corresponding to the current operation
↑	Cursor movement	The program cursor moves up

SPACE 空格		The preceding text is separated from the following text by characters
↓		The program cursor moves down
←		The program cursor moves to the left
→		The program cursor moves to the right
	Turn the page	Display front page
		Display subpage

3.6.3 **Select mode**



Mark	Name	Function
	Start button	Start automatic operation. During automatic operation, the green light will be on.
	Pause button	Abort automatic operation. The indicator lights up when the automatic operation is suspended.
	Safety door open/close	When the door is locked, press the button to open or close the back door. Do not start the program while the safety door is open.
	Program lock	In the “ O ”position, the processing program is protected. The processor cannot be edited while in the protected state. When in the“ | ” position, the processor can be edited
	Emergency stop	If you feel any danger in the process of mechanical operation, press this button will cut off the input power supply on the machine, so that the operation of the machine all stop. In the case of a reset, rotate in the direction of the arrow (clockwise). Then the NC power supply and the main circuit breaker are temporarily OFF and placed ON again.
	Hand wheel	Enter the handwheel movement screen, select the axis to be moved, or the reference point reset.
	Power on button	Switch on the NC power supply.
	Power on button	Disconnect the NC power supply.
	auto	As the program is executed, it is processed in a loop.
	single	After processing a work piece, the program will stop at the program head and a work piece is completed.

Mark	Name	Function
	step	Program completion - does not execute the next program segment.
	mdi	Select it to use the data input key of CNC operating disk to input CNC instructions and execute.
	orign	Select the origin return mode

3.6.4 **Multiplier switch**

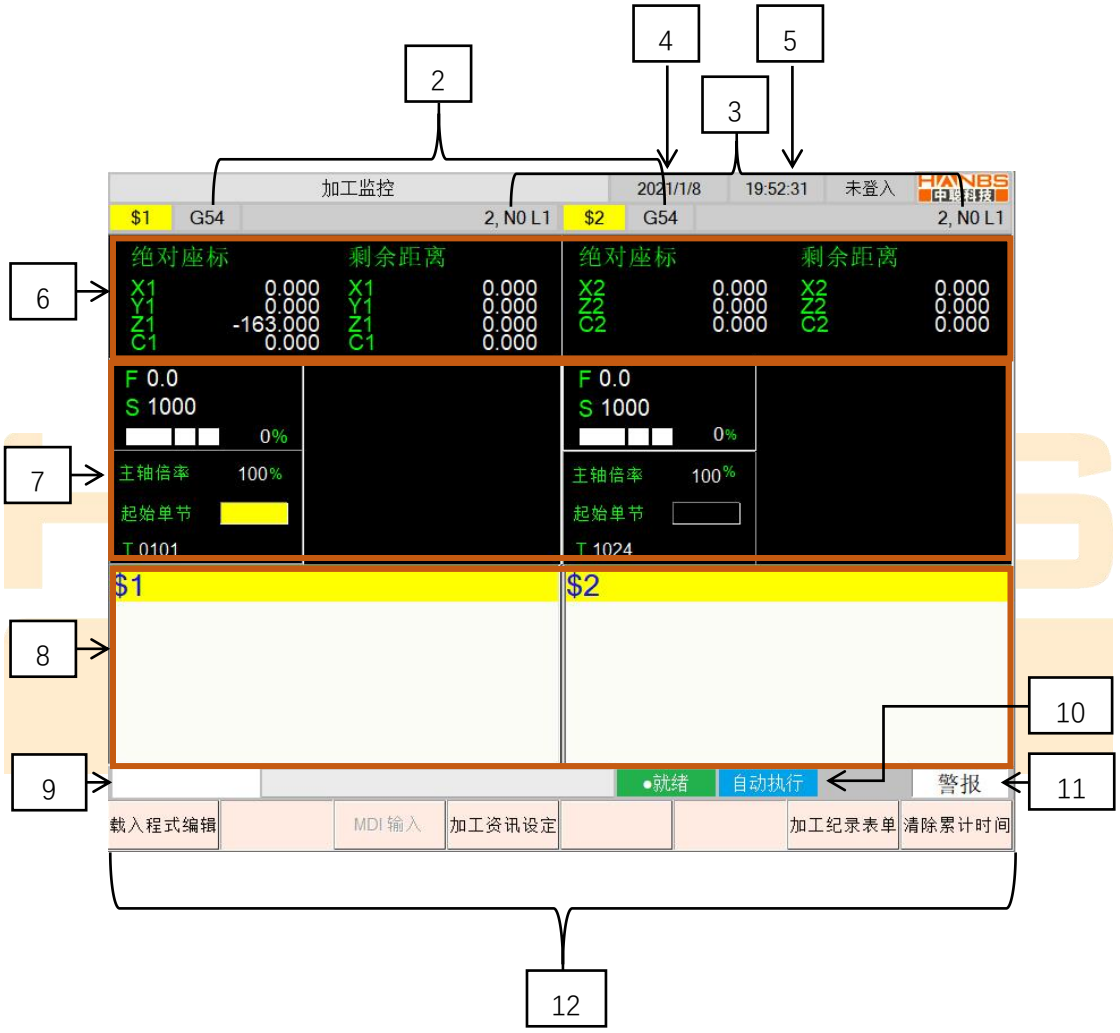
Mark	Name	Function
	Multiplier selection	1. Used when changing the fast forward rate (0 ~ 100%). 2. When changing the manual feed speed (0 ~ 1200mm /min) and cutting feed speed (0 ~ 150%). 3. The set value applies to both fast forward and cutting feed rate. 4. In manual mode (0-8mm/min), the moving distance is 0.001mm (10-100mm/min) Moving distance is 0.01mm (120-1200mm/min) The moving distance is 0.1mm.
		Note:：This switch is a dual switch for setting fast forward speed, cutting feed speed and JOG feed speed. · The upper limit of fast forward rate is 100%. Even if more than 100% is selected, the value is set to 100%.

	Hand wheel simulation	1. In automatic and MDI mode, after opening the handwheel simulation, the movement of the handwheel can be used to control the execution of the program. 2. Hand wheel rotation in the direction of "+" the program will be executed downward, such as hand wheel rotation in the direction of "-" the program will not be executed in the current segment. 3. If the handwheel does not rotate, the program stops and does not execute in the current segment. 4. In manual mode, the axis "X,Y,Z,C1,X2,Z2,C2" can be selected to move in positive and negative directions.
		Note: This function is used for both axial movement in manual mode and execution program in automatic mode. Cutting rate is controlled by multiplier. Moving distance is controlled by multiplier switch

Chapter Ⅳ Function and operation of CNC

4.1Screen switch

4.1.1 screen



⚠ PAY ATTENEION

- Pay attention to keep the screen clean in use.
- Do not use sharp objects and any damage screen objects bump screen to prevent damage.
- The display content will change according to the machine tool state.

No.	Name	Function

No.	Name	Function
1	Channel number	$1 is system 1 and $2 is system 2. You can press  to switch channels .
2	Machining coordinate system	Work piece coordinates set (left $1 System 1, right $2 System 2) G54 is the first working frame. It will default to G54 until a new frame is set.
3	Program number	Example program name set when starting a new program: (O0001).
4	Date	Current date: Year/month/day
5	Time	Current time: hour/minute/second.
6	Absolute coordinates Residual coordinates	The "absolute coordinates" are the current machining location (mm). Residual coordinates Remaining movement of the processed program segment (unit: mm).
7	Processing information monitoring area	This area is overlapped with Processing Information. Use Process Information to switch the display.
8	Program display area	$1 System 1 program. "$2" System 2 program.
9	Data entry field	Enter the data and display the general data number field and data field on the Settings screen.
10	State	Display the current machine status example: when running the program, display "processing", display "ready", etc.
11	Alarm prompt	Alarm prompt icon.
12	Function key selection	The menu is displayed at the bottom of the screen for easy selection and manipulation. According to the location displayed on the menu screen, press the menu key to select the menu.

4.1.2 Main course menu key

The main menu key is used by the operator during the operation and viewing of various functions.

Reference: Function keys for 8 key main menu keys for step configuration. When the destination primary key is not displayed, press the soft key ► (go back to top)

► (The same floor moves to the right)

page	Main course menu function key	备注
1	机台坐标 | 程序编辑 | 刀具管理 | 加工监控 | 维护 | 用户参数 | 功能设置 | 辅助面板	
2	PLC 状态 | 诊断功能 | 参数设定 | 系统管理 | | | | 系统信息	

4.2 Processing information settings

Reference:

1. This area overlapped with "Processing Settings". Continuously press the key "加工资讯设定" Processing display area, program code monitoring area, processing information Display area" to switch back and forth.
2. You can use this page to view and modify relevant data.
3. The machine monitoring display screen is the machining program "cutting parameters", and the machine moving position display, program state monitoring.
4. Start single section 起始单节 can be operated by pressing ◄ ►, other information can not be operated on it, only display.

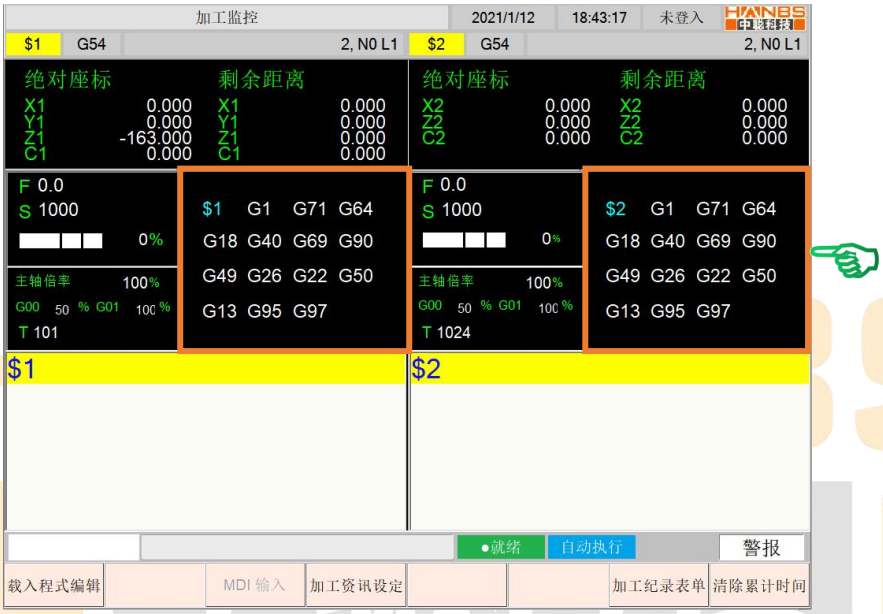
4.2.1 **Machine monitoring area**



Operation	Name	Function
	F	Cutting feed speed monitoring（m/min）
	S	Main/auxiliary shaft speed monitoring（r/min）
	%	Percentage of machine machining feed speed
	Principal axis ratio	Spindle magnification shows percentage
加工资讯设定	Start a single section	n: The processing start section can be specified as N lines (EX: N20) L+n: Specify the start single section as line N (EX: L20) N+n: Searches for the line number of N+n and specifies the starting line of the action (EX: N3) T+n: Searches for the row number of T+n and specifies the start row of the change action (EX: T01) Specifies the last line of a program if the specified line number exceeds the program maximum number of lines
	T	T-code display example: T0402 The first two digits of 04 are knife numbers displayed in the current execution The last two digits are 02. The knife complement is displayed during the current execution

4.2.2 程序码监控区

Reference:	Page for G code mode monitoring. When the program processing G code changes, the page status will change accordingly. The code that is not kept when the Reset key is pressed will be cleared automatically.



Operation	Name	Function
加工资讯设定	Code monitoring area	This area displays the current code usage status. Use "Processing Information Settings" to switch the display

4.2.3 加工资讯显示区

Reference:	This area overlaps with "processing Settings", and is displayed by pressing the button 加工资讯设定 "continuously. “ The left area is editable. The right area is not editable. Number of jobs When the number of jobs reaches the specified M code (parameter #3804 can specify M code), the number of

jobs increases by 1. When the number of jobs reaches the required number, the console automatically enters the suspended state.

Yellow cursor move Press the keyboard ↑ ↓ to move, and the value in the yellow cursor can be modified.





Operation	No	Name	Function
	1	Accumulated processing time	Machine current accumulated processing time: (cannot be modified, only display)
		Processing time	Current program processing time: (cannot be modified, only display)
		Breakpoint line number	Show the line number when the program was last interrupted: (cannot be modified, show only)

加工资讯设定		Breakpoint number	Show the sequence number when the program was last interrupted: (cannot be modified, only show)
	2	Number of work pieces	The number of machines processed at present Note: The number of workpieces will be cleared when the processing file is changed
		Number of required jobs	Set the total number of workpieces to be processed When the number of jobs reaches the Number of Jobs Required, the platform will enter the pause state
		Total number of jobs	Machine total processing several pieces The system cannot automatically return to zero, only the user manually return to zero
		Feed rate	Speed when the front machine is moving
		Spindle speed	Current spindle speed display
	3	Spindle state	The first spindle: the main shaft The second main axis: the back axis Third main shaft: main shaft side milling power head The fourth spindle: auxiliary shaft end milling power head Other spindles are not in use
		multiplier	Current magnification monitoring of each spindle（%）
		Spindle speed	Current spindle speed monitoring (for example, 1000RPM is displayed, but the actual speed is 1000RPM) If 0, the spindle is stopped
		Load rate	Load monitoring of each axle (%) Used to check whether the axis rotation is smooth Excessive load during turning or milling may damage the spindle coil

4.3 Processing record form

Reference:

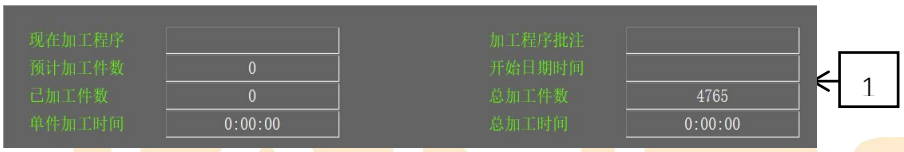
Press " 加工资讯设定 " and then " 加工纪录表单 " to display the processing record form screen.

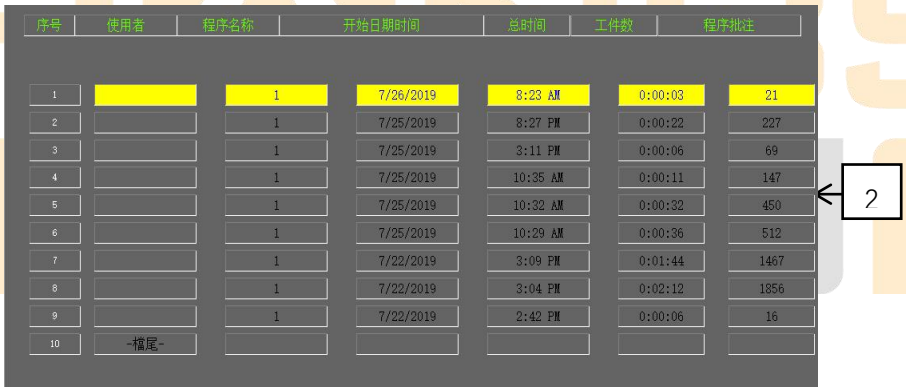
The record form records the last 10 processor forms.

The current processing record table can be viewed, and the worker will remit the processing record to the external magnetic spy machine.

Press the keyboard key ↑ ↓ Move the yellow cursor.



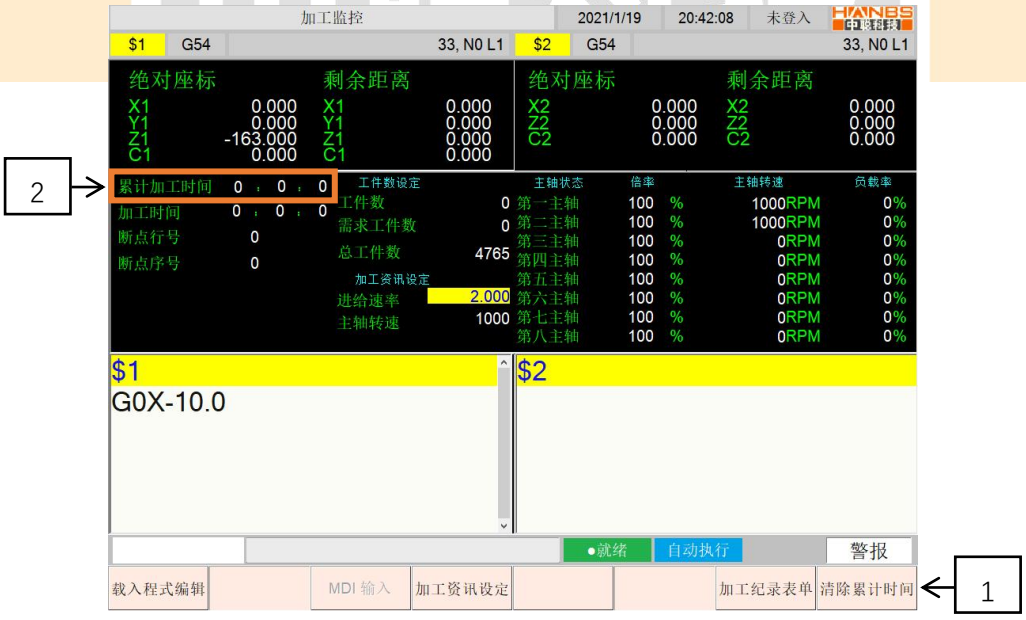




Operation	No	Name	Function
加工纪录表单	1	Current processing program	Name of the current loader
		Expected number of processed pieces	Same as the number of required jobs
		Number of processed pieces	Current number of working pieces
		Single piece processing time	The current program loops for one time
		Process annotation	Program annotation ($1,$2 if no annotation)
		Start date and time	The date the program loaded the current

Operation	No	Name	Function
			processing time
		Total number of processed pieces	Total processing quantity is shown
		Total processing time	Total processing time of the current program
	2	Program form display area	The last 10 programs loaded the processing form Store up to 10 program records
	Storage processing record		To export the current processing record form outside (U disk and diskette drive), press F1 "Store plus record", select the location to export ↑ ↓ select device, press keyboard ENTER 回车 key or press F1 "is" OK ".
	Clear machining record		To delete the current record form, press F2 "Clear Processing Records" and press F1 "is" Record Clear ".

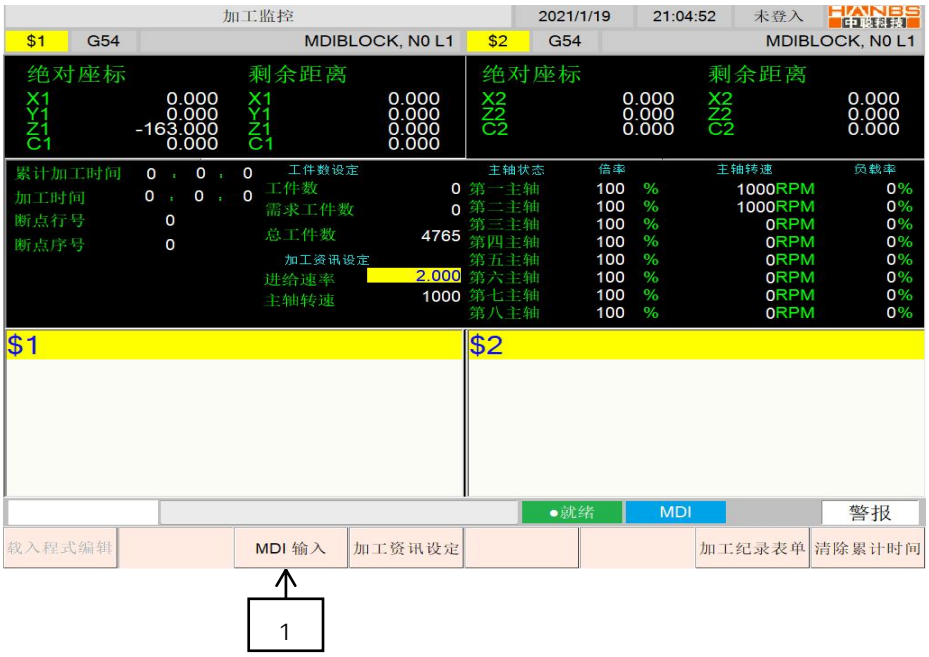
4.4 Clean processing time

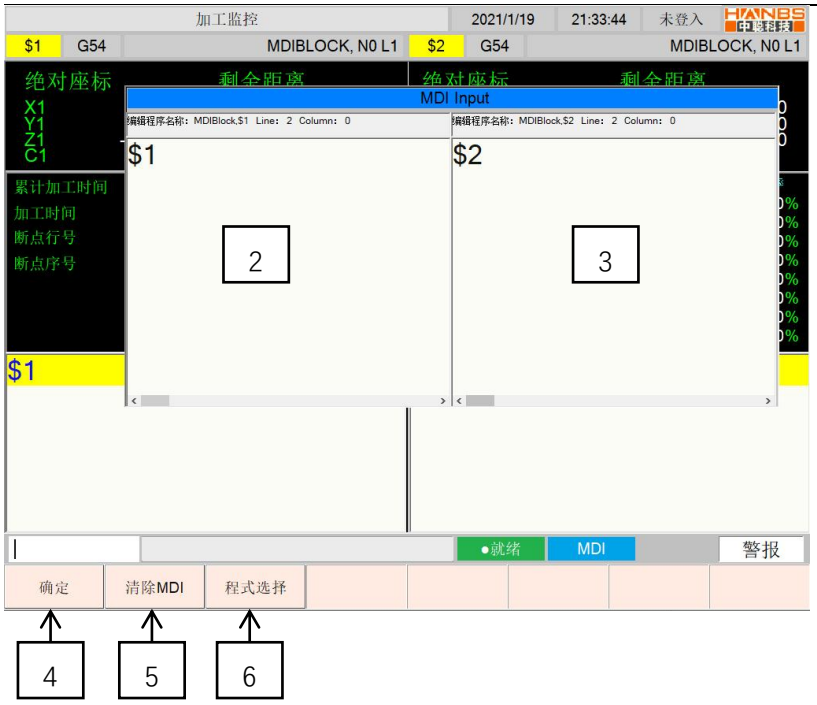


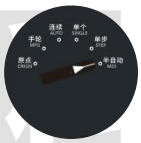
Operation	No	Name	功能
清除累计时间	1	Clear accumulated time	To clear the total time of the current processing program, press F8 " 清除累计时间 " to display  screens, and then press F1 是 to clear the "cumulative processing time" to zero.
	2	Accumulated processing time	After the operation of the first step above, the accumulated processing time is cleared to zero.

4.5 MDI Enter

⚠WARNING	Ensure the machine and tool are in a safe position when executing MDI procedure. When executing MDI program, the key must be lit to prevent misoperation. Ensure that safety gates are closed when performing MDI procedures to prevent personal injury.





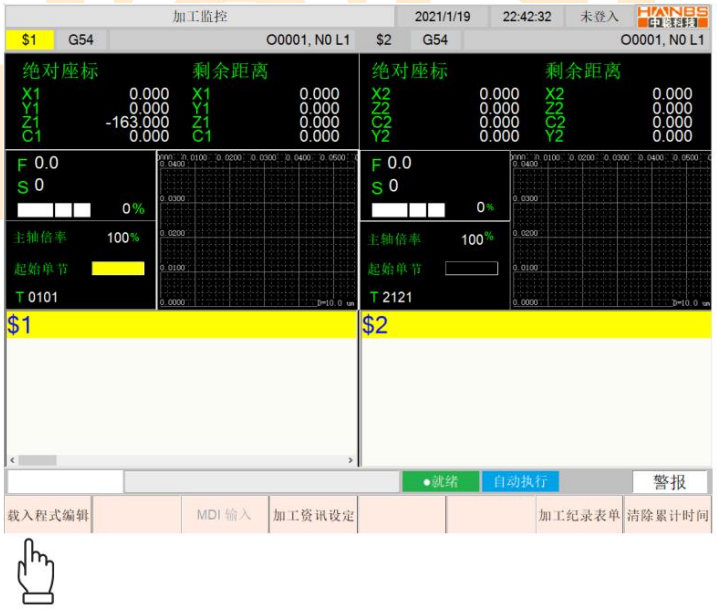
Operation	No	Name	Function
	1	MDI enter	MDI input button. Note: Before editing MDI, ensure that the band switch is in the "MDI semi-automatic" gear  ; otherwise, the MDI input key will be gray and cannot be entered.
	2	$1 Spindle editing area	Spindle MDI program editing area Move the cursor with the ↑ ↓ ← → key on the keyboard
	3	$2 Auxiliary axis edit area	Accessory axis MDI program editing area Move the cursor with the ↑ ↓ ← → keyboard keys
	4	Confirm button	The program can be executed after the "confirm" key is completed
	5	Clearing MDI	To clear the contents in MDI, press F2 "Clear MDI" to clear all contents in MDI (including the contents in $1 and $2). To delete a single item, press CAN 删除 on the keyboard

Operation	No	Name	Function
	6	Program selection	The $1 spindle editing area switches with the $2 auxiliary axis editing area When the "black cursor" flashes in that area, it is the current editing area

4.6 Program editing

4.6.1 Loader program edit

⚠ WARNING	Ensure the machine and tool are in a safe position when executing the procedure. The mode is in the automatic execution state. The key  must be lit when executing the program to prevent misoperation. Ensure that safety doors are closed when performing procedures to prevent personal injury.



Operation	Name	功能
载入程式编辑	Loading program processing	Press the soft key ◀ ▶ to return to the main screen, press F4 加工监控 to enter the above

Operation	Name	功能
		screen, and then press F1 [载入程式编辑] to enter the program editing page. If the screen is on the main screen, just press F1[载入程式编辑]。

4.6.2 (G,M,T) code screen list

Reference:	A list of available G and M codes can be found on the "Program Edit" page This page can only be referenced and cannot be edited







【 Operation method 】

1. Press the soft key ◄ ► to find the main page

2. Press F4 Processing monitor 加工监控 Enter the processing screen

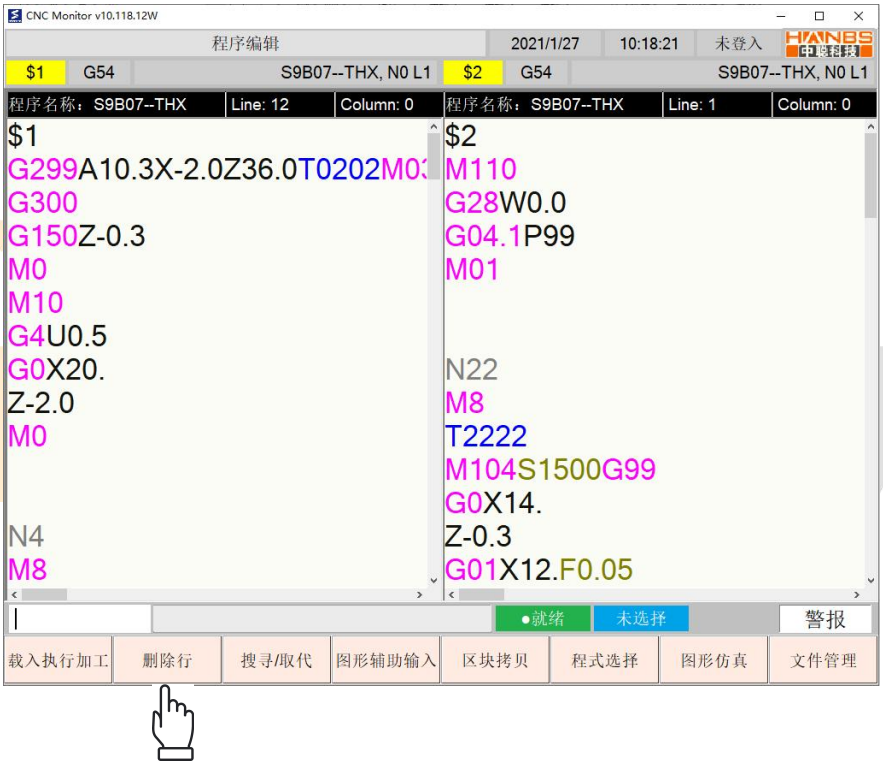
3. Press the F1 载入程式编辑 program edit button

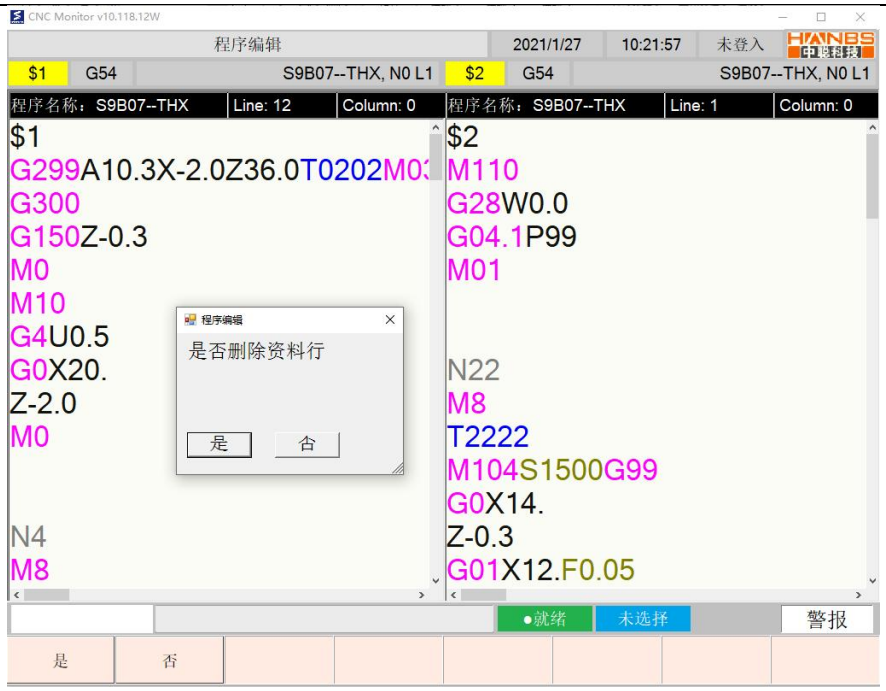
4. Press the keyboard HELP key to display G,M,T code screen

5. Press the page turning key on the keyboard to switch the display

4.6.3 **Delete line**

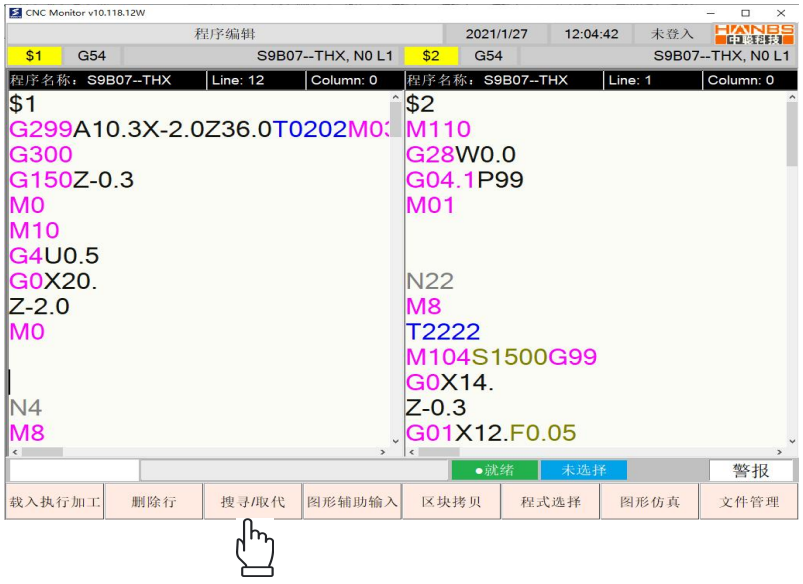
⚠ WARNING	Before deleting the program, please make sure that the content you want to delete cannot be restored once deleted. Program header $1,$2 cannot be deleted. After deleting the program, please check the program content to prevent personal safety accidents caused by errors.

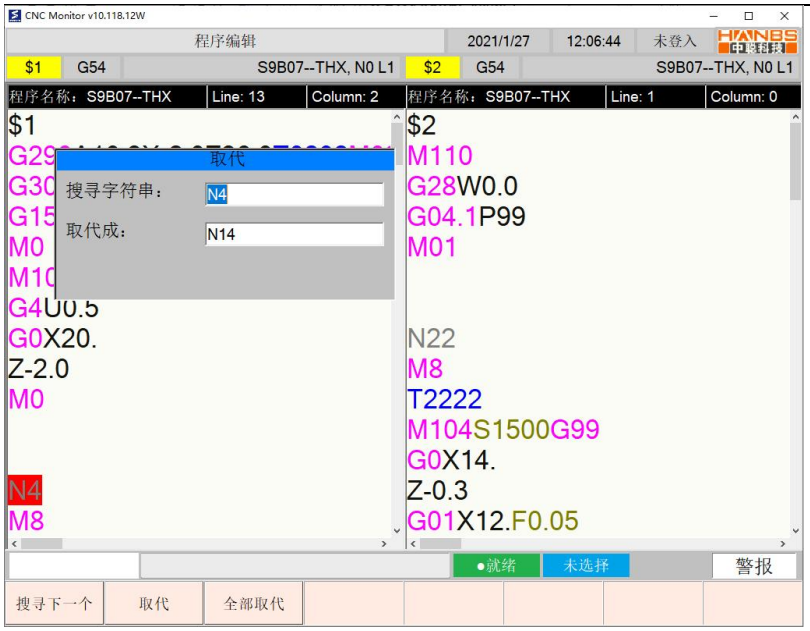




Operation	Name	功能
删除行	Erase program	Move the cursor to the position where the program needs to be deleted and press F2 删除行 screen to show whether to delete the program. If not, press F2 (No), and if to delete, press F1 (Yes).

4.6.4 **Search for/replace**





【 Operation method 】

1. Press the soft key ◀ ▶ to find the main page

2. Press F4 加工监控 processing monitor to enter the processing screen

3. Press the F1 载入程式编辑 program edit button

4. Press the F3 搜寻/取代

5. 5. At this time, a dialog box will be displayed (as shown in Figure 2).

6. Press the keyboard ↑ ↓ key to move the cursor to the Search String text box

7. Enter the name of the byte to be searched in the input box such as (N4)

8. Press F1 key "Search Next" on the display screen. At this time, the program will automatically search down for N4. As shown in Figure 2, the searched N4 will be covered with red

9. Press the keyboard ↑ ↓ key to move the cursor to the "Replace" text box

10. In the Replace To text box, enter the byte you want to replace

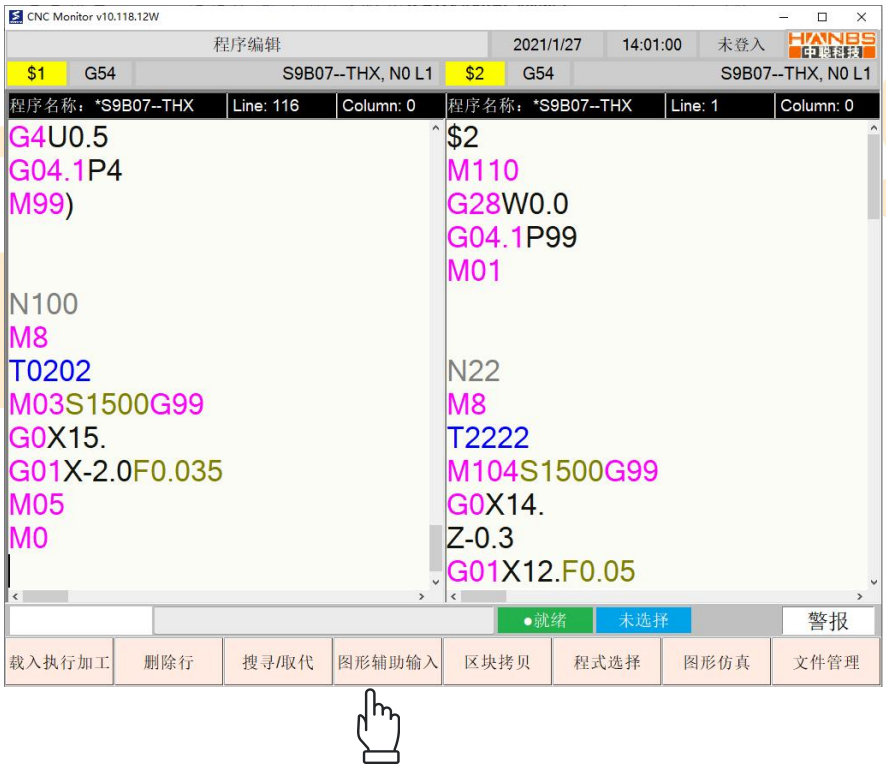
11. Press F2 取代 to complete the replacement. If there are multiple N4s in the program, you can also press F3 全部取代 to replace all the characters

4.6.5 Graphics assisted input

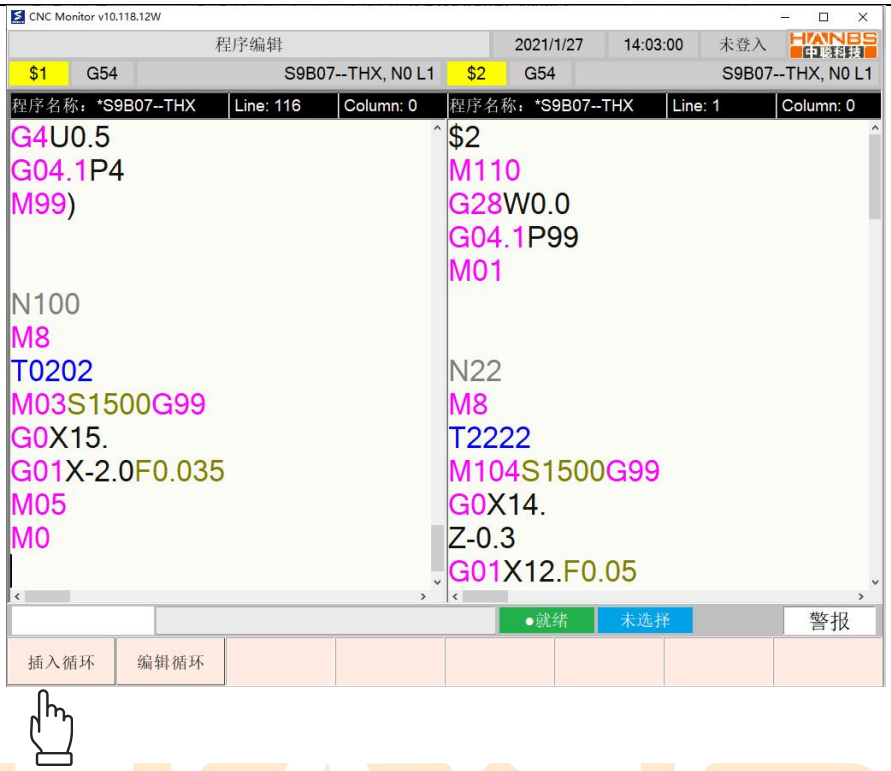
4.6.5.1Insert loop

Reference:	"Graphics auxiliary input" function is designed for beginners to develop a graphic input dialog box, through the input box to fill in the value of the system automatically generated processing program. "Graphics assisted input" menu is divided into two sub-menus "Insert loop" "Edit loop" two, two menus are divided into several sub-menus. Please check the program carefully after editing to prevent errors. Make sure the machine is in the Ready state before editing the program.

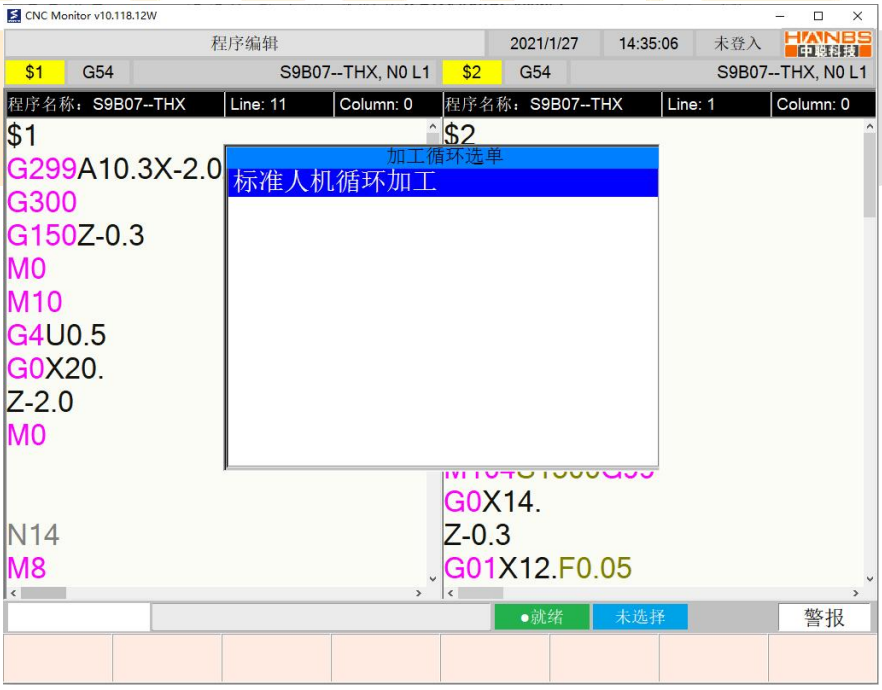
（图 1）



（图 2）



（图 3）



【 Operation method 】

1. Press the soft key ◄ ► to return to the main page.

2. Press F4 Processing Monitor 加工监控 to access the processing page.

3. Press the F1 载入程式编辑 program edit button.

4. Use keyboard ↑ ↓ ← → to move the cursor to the programming position.

5. Press F4 图形辅助输入.

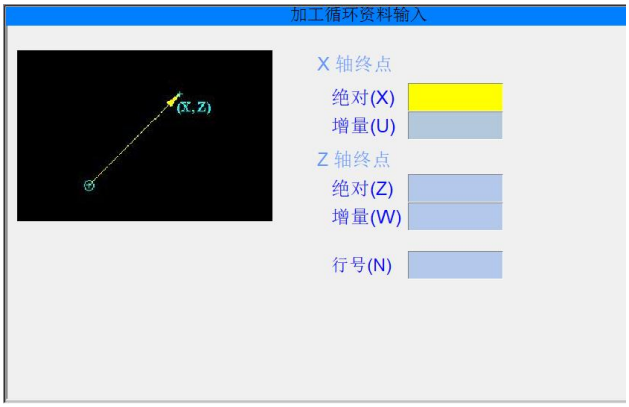
6. Press F1 插入循环, and "Standard man-machine cycle processing" will appear in the main screen (Figure 3).

7. Press the keyboard ENTER 回车 key and the following function selection will appear in the main screen.



8. Move the cursor up and down the keyboard ↑ ↓ to select the G-code program to be generated automatically.

9. Press the keyboard ENTER 回车 key to enter the graphics input options page as shown in the following figure.



10. Enter the corresponding value in the corresponding box and press ENTER 回车 or F1 "OK" to complete.

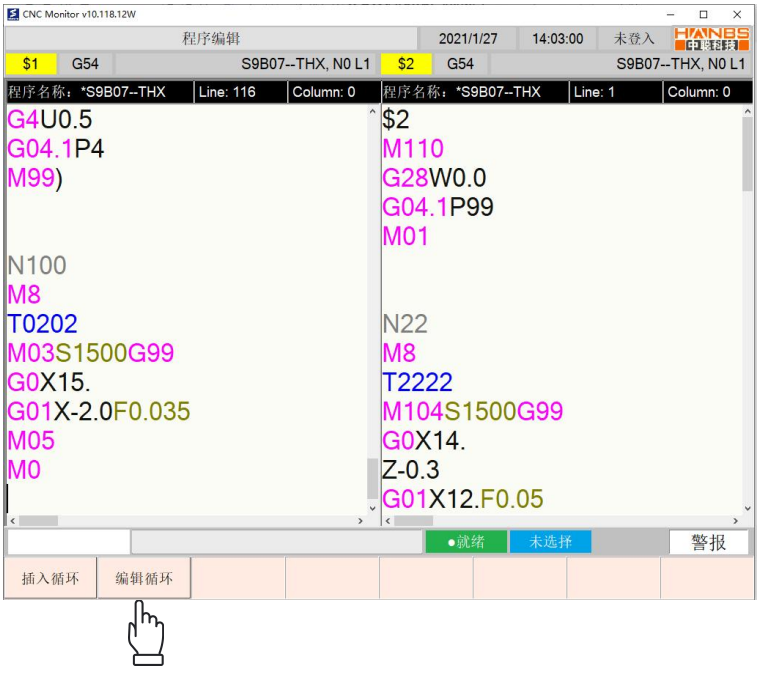
4.6.5.2Edit loop

Reference:	The function of "edit loop" is that the system automatically generates the processing program through man-machine dialogue according to the G code written in advance by the programmer. Please check the program carefully after editing to prevent errors. Make sure the machine is in the Ready state before editing the program.

(图 1)



(图 2)



(图 3)



【 Operation method 】

1. Press the soft key ◀ ▶ to find the main page.

2. Press F4 processing monitor 加工监控 to enter the processing screen.

3. Press F1 载入程式编辑 program Edit key.

4. Move the cursor to the programming position through the keyboard ↑ ↓ ← → and enter the loop code G75 that needs to automatically generate the program.

5. Press the F2 编辑循环 button to display the screen (Figure 3).

6. Use keyboard ↑ ↓ ← → to move the cursor to the corresponding box and enter a value.

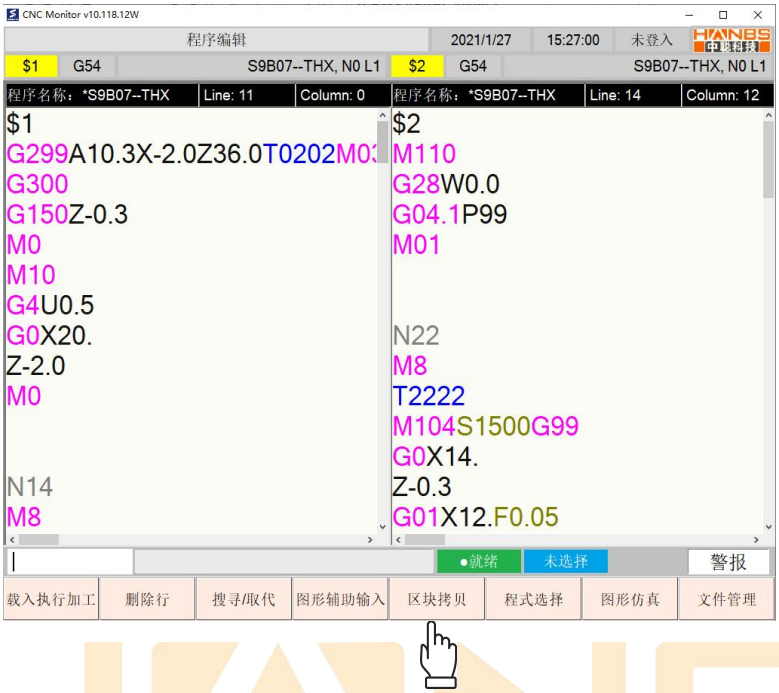
7. 7. Press F1 "OK" or press the keyboard ENTER 回车 to complete the input.

4.6.5.3 Block copy

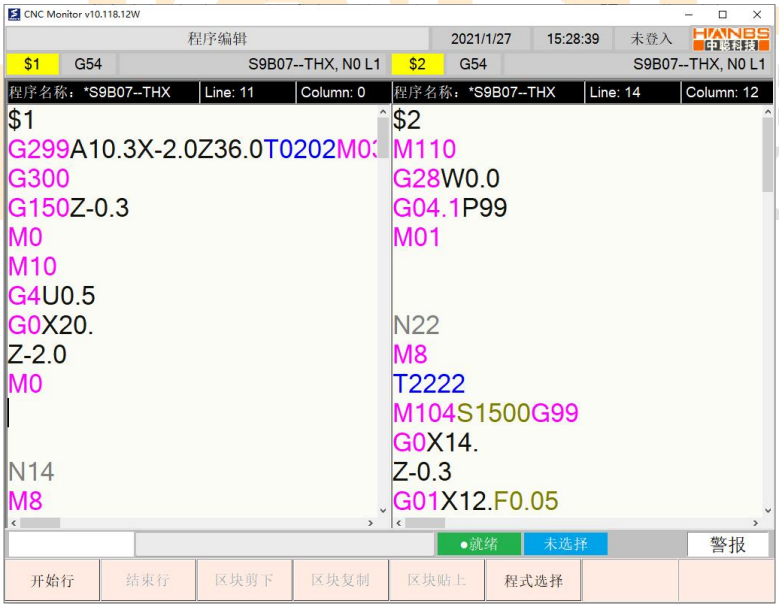
Reference:	"Block copy" function is the system according to the pre-written G code through the man-machine dialogue automatically generated processing program. Please check the program carefully after editing to prevent errors. Make sure the machine is in the Ready state before editing the

program.

（图 1）



（图 2）



【 Operation method 】

1. Press the soft key ◀ ▶ to return to the main page

2. Press the soft key F4 Processing Monitor 加工监控 to access the processing page.

3. Press F1 载入程式编辑 program Edit key.

4. Move the cursor to the start position on keyboard ↑ ↓ ← →.

5. Press F1 开始行.

6. Move the cursor to the end position on keyboard ↑ ↓ ← →.

7. Press F2 结束行。

8. Press F3 区块剪下 or F4 区块复制 select the desired functionality.

9. Move the cursor to the position you want to paste using keyboard ↑ ↓ ← →.

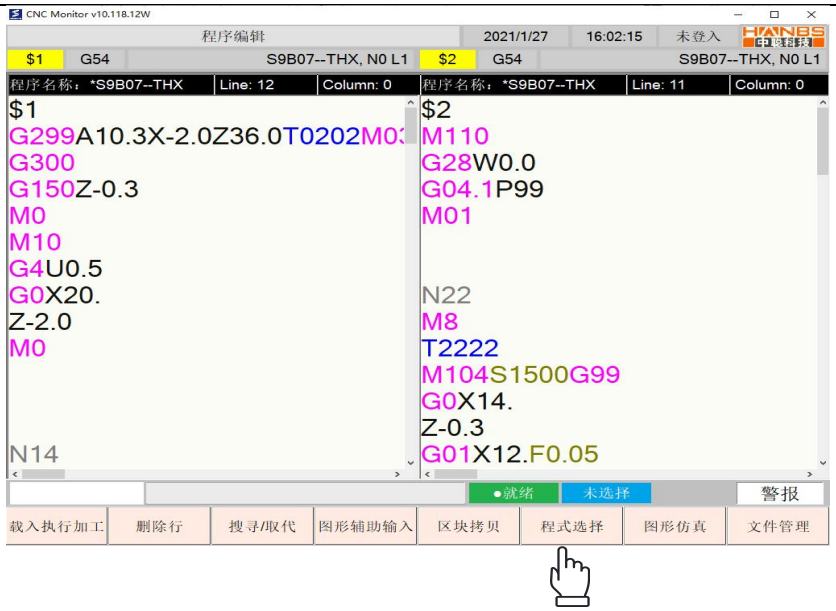
10. Press F5 区块贴上 to complete the copy.

11. 11. Switch between System 1 ($1) and System 2 ($2) by pressing F6 程式选择.

4.6.5.3 Program selection

Reference:	The "Program selection" function switches between system $1 and system $2 editing areas 2. Check whether the cursor is displayed in system 1 and system 2 3. Make sure the machine is in the Ready state before editing the program

（图 1）



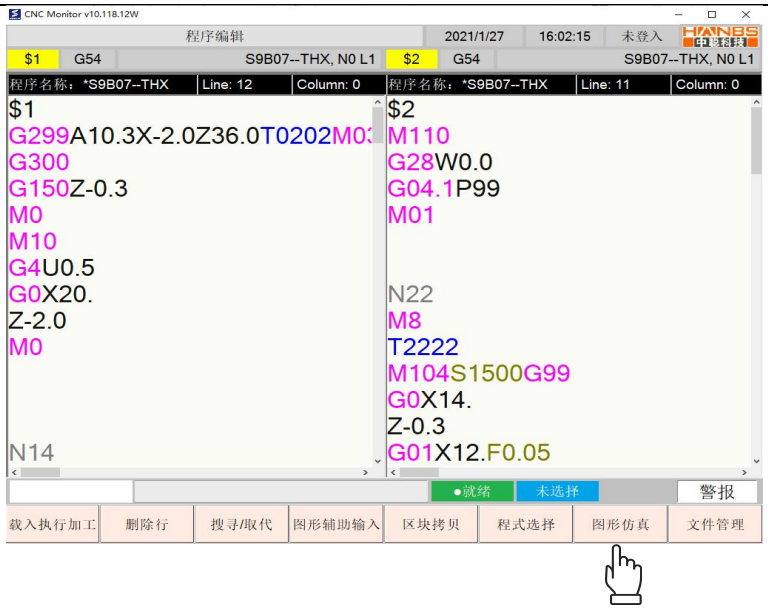
【 Operation method 】

1. Press the soft key ◀ ▶ to return to the main page.
2. Press F4 processing monitor 加工监控 to enter the processing screen.
3. Press the F1 载入程式编辑 program edit button (Figure 1).
4. Press F6 程式选择 to switch between System $1 and System $2.

4.6.5.4 Graphic truth-proof

Reference:	"Graphics anti - true" function is to check whether the program editing error or not Switch between system $1 and System $2 graphic anti-authenticity through "Program Selection" in section 4.5.6 3. Make sure the machine is in the Ready state before editing the program

（图 1）



（图 2）



【 Operation method 】

1. Press the soft key ◀ ▶ to return to the main page

2. Press the soft key F4 processing monitor 加工监控 to enter the processing screen.

3. Press the F1 载入程式编辑 program edit button.

4. Press F7 图形仿真 (Figure 1).

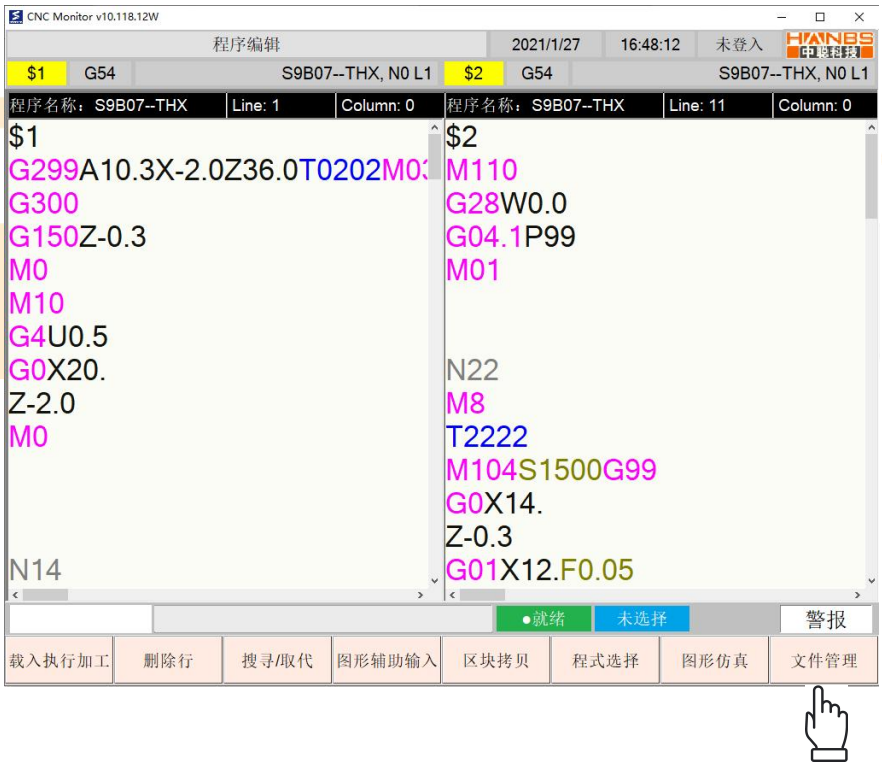
5. At this time, it is displayed (as shown in Figure 2) that the operator uses the soft keys F1,F2,F3,F4,F5,F6 to enlarge and shrink the analog figure.

4.7 File management

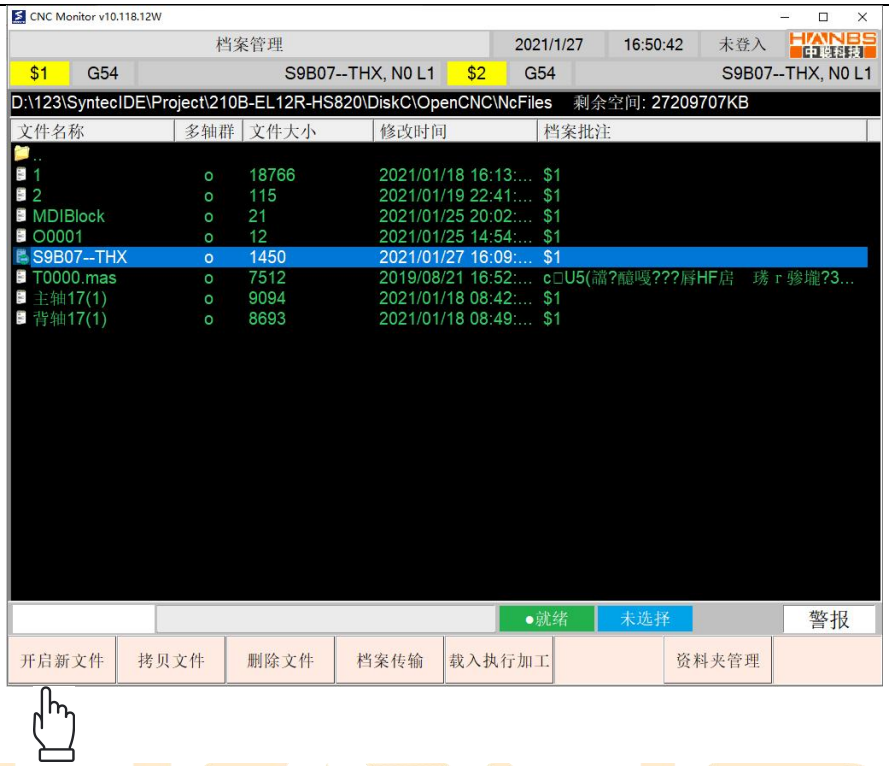
4.7.1 Opening a new file

⚠ WARNIN G	If the machine is running the program to open the new file belongs to the background editing, must not press the button "load execution processing" to prevent the processing program and machining tool inconsistent resulting in collision. It is recommended to name new files with letters and numbers. After opening the new file, please check more procedures to prevent the error caused by collision. The open file name cannot be O9980-O9999 because internal system programs have been used. Please note

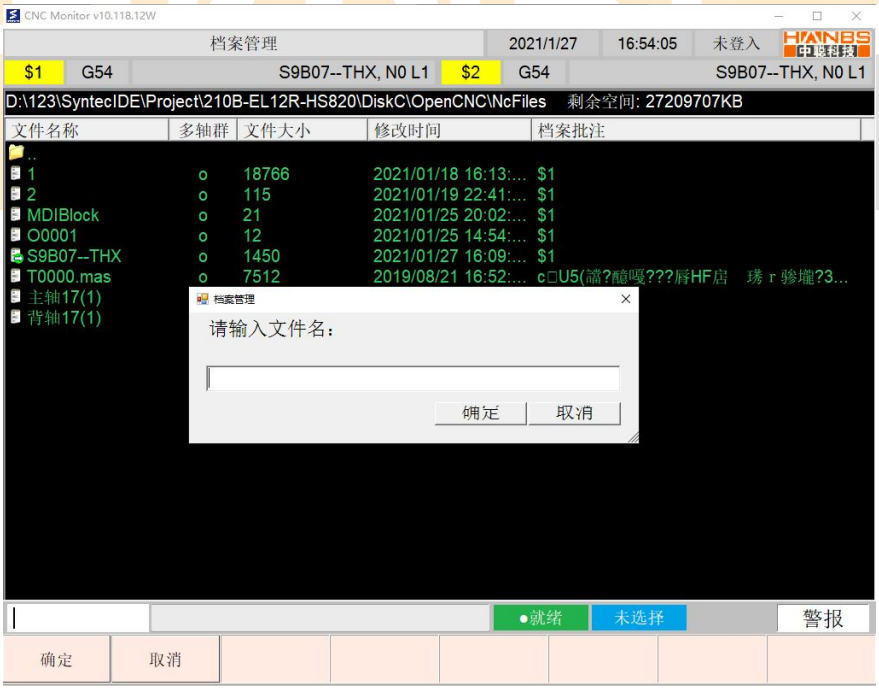
（图 1）



（图 2）



（图 3）



【 Operation method 】

1. Press the soft key ◄ ► to return to the main page

2. Press the soft key F4 Processing Monitor 加工监控 to access the processing page.

3. Press the F1 载入程式编辑 program edit button.

4. Press F8 文件管理 (Figure 1).

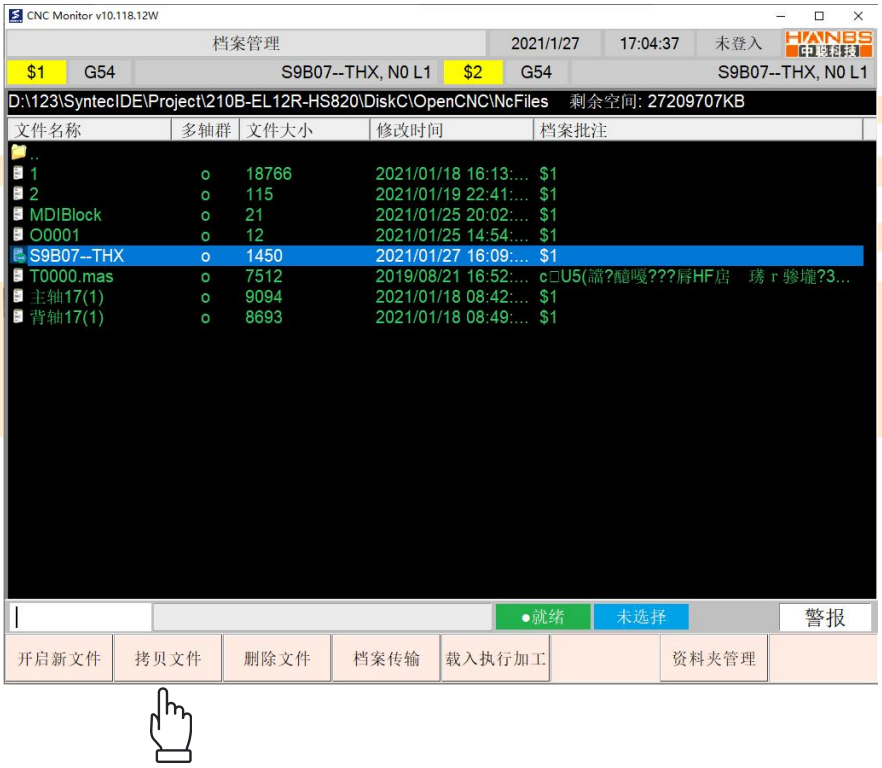
5. Press F1 开启新文件 to open a new file (Figure 2).

6. Enter the name of the file to open (Figure 3).

7. Press F1 "Confirm" to open the new gear.

4.7.2 **Copy file**

（图 1）



（图 2）

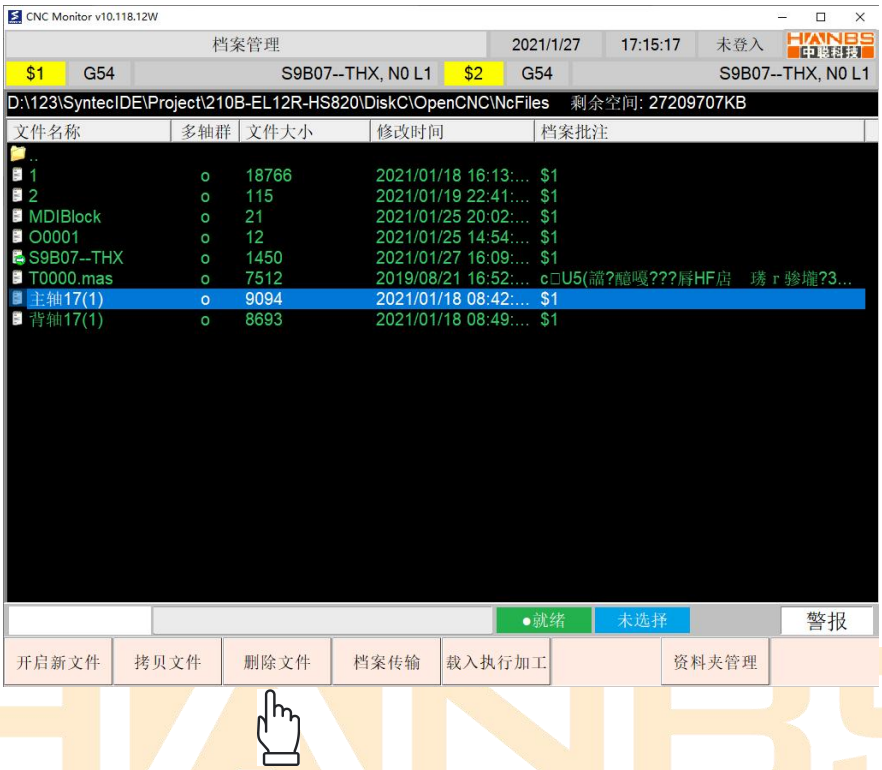


【 Operation method 】

1. Press the soft key ◀ ▶ to return to the main page .
2. Press the soft key F4 Processing Monitor 加工监控 to access the processing page. 3
3. .Press the F1 载入程式编辑 program edit button.
4. 4.Press F8 文件管理 (Figure 1).
5. Use the keyboard ↑ ↓ to move the cursor to the file you want to copy.
6. Press F2 拷贝文件 to copy the file (Figure 1).
7. Enter the new file name (Figure 2).
8. Press F1 "Confirm" to copy file complete.

4.7.3 **Deleting file**

（图 1）



（图 2）



【 Operation method 】

1. Press the soft key ◄ ► to return to the main page.

2. Press F4 processing monitor 加工监控 to enter the processing screen.

3. Press the F1 载入程式编辑 program edit button.

4. Press F8 文件管理 。

5. Press F3 删除文件 See (Figure 1).

6. The screen shows (Figure 2) moving the cursor with keyboard ↑ ↓ to the location where you want to delete the file.

7. Press F1 选择. The small box in front of ☑ the file name is selected.

8. Press F2 选择所有 to select all documents.

9. To deselect the F3 取消选择 key, the small box at the front of the file name is ☑.

10. Press F4 删除档案 to delete files ☑ .

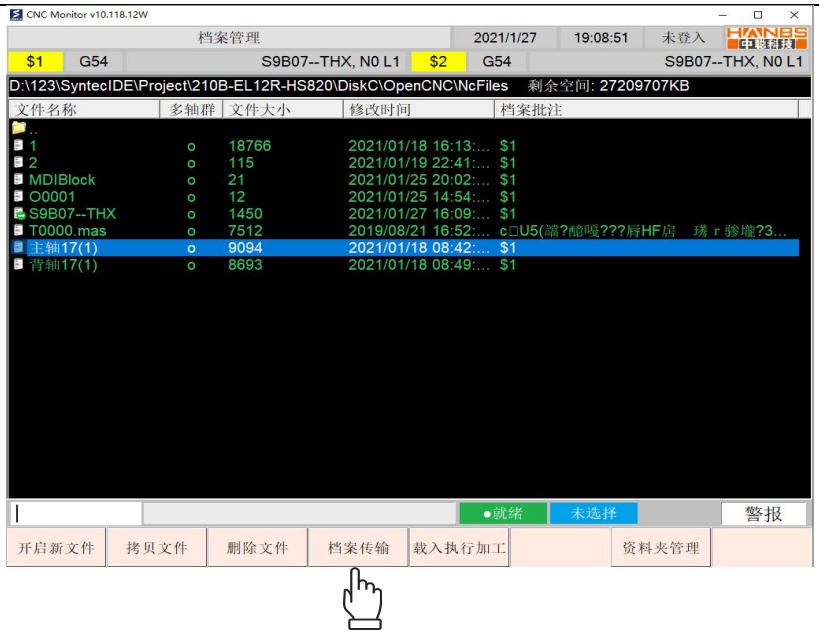
11. To delete all files, press F5 删除全部档案.

4.8 File transfer

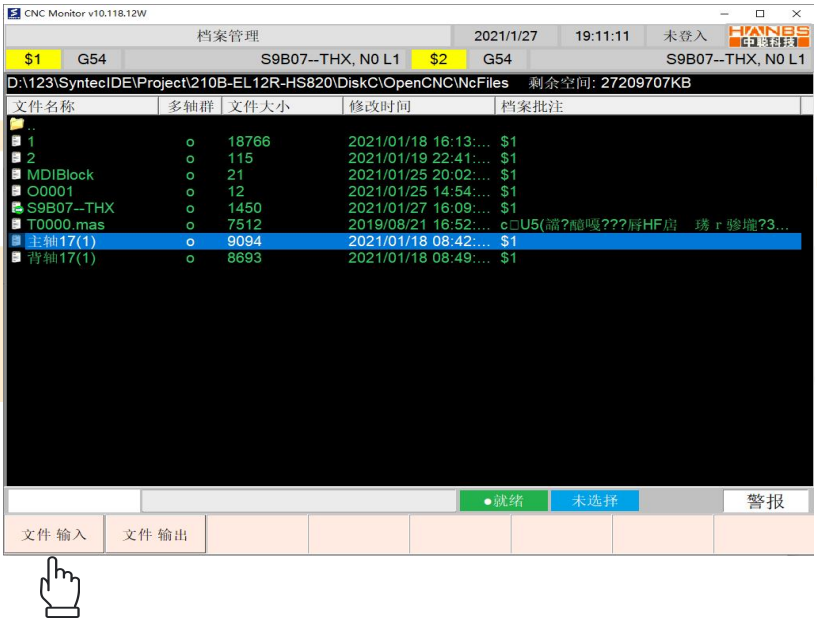
4.8.1 File input

Reference:	"File input" refers to the transfer of programs from external USB flash drives including CF cards to machine tools The input file should not be too large It is recommended that operators develop good program management habits and copy the non-processing program to the U disk or CF card The suffix of the input file is deleted. For example, O1111 is correct

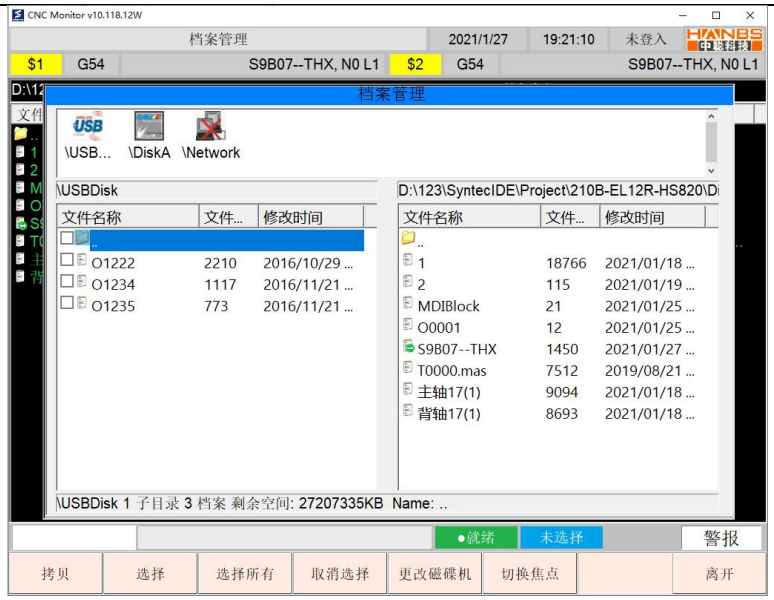
（图 1）


CNC Monitor v10.118.12W
档案管理
2021/1/27
19:08:51
未登入
$1 G54 S9B07--THX, N0 L1 $2 G54 S9B07--THX, N0 L1
D:\123\SyntecIDE\Project\210B-EL12R-HS820\DiskC\OpenCNC\NcFiles 剩余空间: 27209707KB
文件名称 多轴群 文件大小 修改时间 档案批注
..
1 o 18766 2021/01/18 16:13:... $1
2 o 115 2021/01/19 22:41:... $1
MDIBlock o 21 2021/01/25 20:02:... $1
O0001 o 12 2021/01/25 14:54:... $1
S9B07--THX o 1450 2021/01/27 16:09:... $1
T0000.mas o 7512 2019/08/21 16:52:... c□U5(譜?醷嘎???脣HF店 璣ｒ骾瓏?3...
主轴17(1) o 9094 2021/01/18 08:42:... $1
背轴17(1) o 8693 2021/01/18 08:49:... $1
●就绪
未选择
警报
开启新文件
拷贝文件
删除文件
档案传输
载入执行加工
资料夹管理

(图 2)


CNC Monitor v10.118.12W
档案管理
2021/1/27
19:11:11
未登入
$1 G54 S9B07--THX, N0 L1 $2 G54 S9B07--THX, N0 L1
D:\123\SyntecIDE\Project\210B-EL12R-HS820\DiskC\OpenCNC\NcFiles 剩余空间: 27209707KB
文件名称 多轴群 文件大小 修改时间 档案批注
..
1 o 18766 2021/01/18 16:13:... $1
2 o 115 2021/01/19 22:41:... $1
MDIBlock o 21 2021/01/25 20:02:... $1
O0001 o 12 2021/01/25 14:54:... $1
S9B07--THX o 1450 2021/01/27 16:09:... $1
T0000.mas o 7512 2019/08/21 16:52:... c□U5(譜?醷嘎???脣HF店 璣ｒ骾瓏?3...
主轴17(1) o 9094 2021/01/18 08:42:... $1
背轴17(1) o 8693 2021/01/18 08:49:... $1
●就绪
未选择
警报
文件输入
文件输出

（图 3）



【 Operation method 】

1. Press the soft key ◀ ▶ to return to the main page.

2. Press F4 processing monitor 加工监控 to enter the processing screen.

3. Press the F1 载入程式编辑 program edit button.

4. Press F8 文件管理.

5. Press the F4 key 档案传输 (Figure 1).

6. The screen shows (Figure 2) pressing the F1 文件 输入 key.

7. Press the keyboard ENTER 回车 key to confirm the following figure (shows USB device).

8. Press the keyboard ↑ ↓ to move the cursor to select the file you want to enter.

9. Press F2 选择 to select the file to be copied or press the keyboard ENTER 回车 to select the program to be input（☑for the program to be input）.

10. To select all files at once press F3 选择所有 to select all.

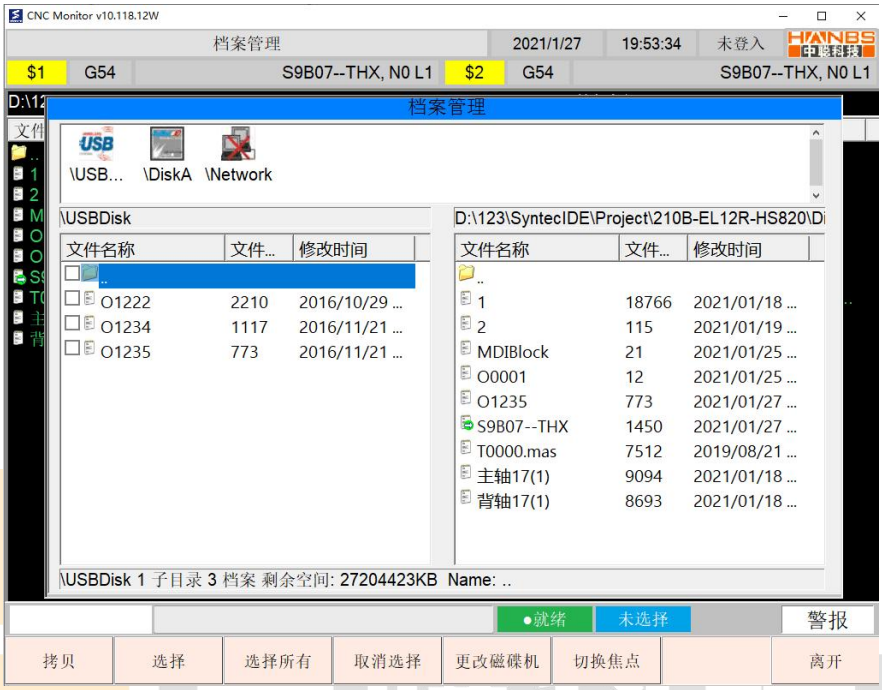
11. Press F1 拷贝 to complete the copy.

12. Press F8 离开 to leave the screen.

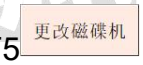
⚠ PAY ATTENTION

➢ To change the drive, do as follows

（图 1）



【 Operation method 】

1. To change the drive, press F5 

。

2. Press the keyboard  to move the cursor to select the disk drive to be changed as shown in the following figure (the icon is gray for the selected device).



3. Click OK on the keyboard 

.

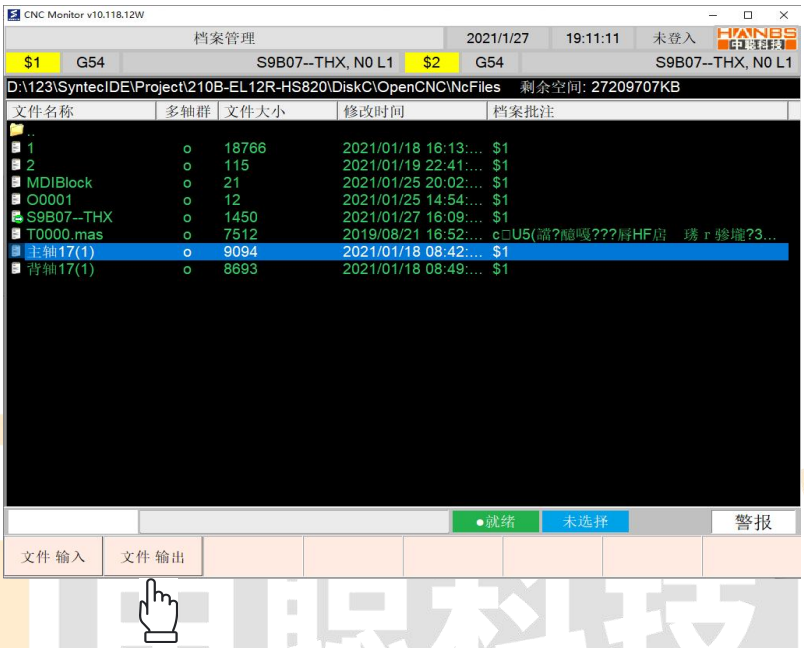
4. Press F8 

to leave the screen.

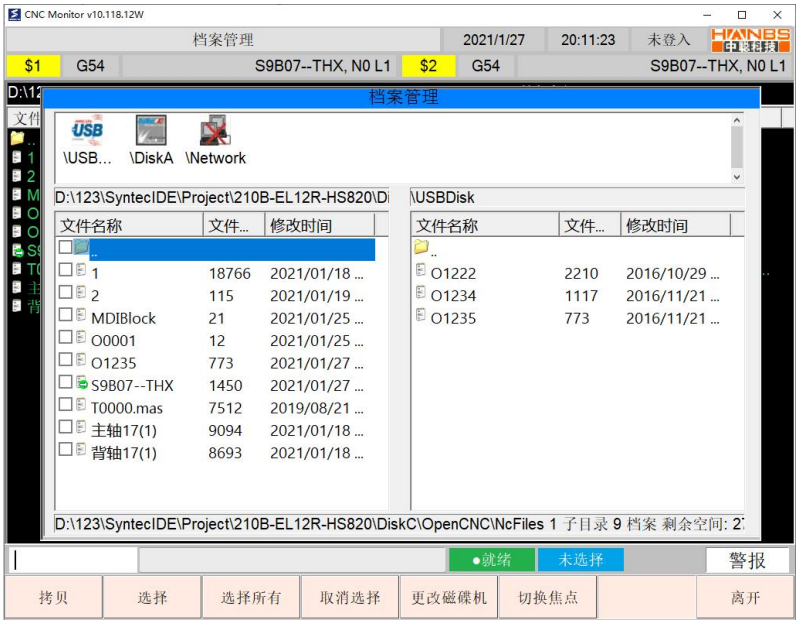
4.8.2 File output

Reference:	"File output" refers to the transfer from inside the machine tool to an external USB flash drive, including the CF card The method of File Output is the same as that of File Input

（图 1）



（图 2）



【 Operation method 】

1. Press the soft key ◀ ▶ to return to the main page.
2. Press F4 processing monitor 加工监控 to enter the processing screen.
3. Press the F1 载入程式编辑 program edit button.
4. Press F8 文件管理.
5. Press F4 档案传输 (Figure 1).
6. 6. Screen display (Figure 2) Press the F2 文件 输出 key.
7. Press the keyboard ENTER 回车 key to confirm the following figure (USB device displayed).
8. Press the keyboard ↑ ↓ to move the cursor to select the file to output.
9. Press F2 选择 to select the file to be copied or press the keyboard ENTER 回车 to select the program to be input (☑ for the program to be input).
10. To select all files at once, press F3 选择所有 to select all files.
11. Press F1 拷贝 to complete the copy.
12. Press F8 离开 to leave the screen.

4.9 Frame coordinates

Reference:	Console Coordinates displays the various coordinates of the current position The coordinate system display can be switched by the soft keyboard F1-F8



Operation	No	Name	Function
机台坐标	1	**Coordinate switching**	Press F1 "Coordinate switch" to display mechanical coordinates, relative coordinates, absolute coordinates and remaining distance display switch.
	2	**1/2 coordinate**	"1/2 coordinates" are only valid for relative coordinates
	3	**Relative coordinate clearing**	Switch to the relative coordinates display screen and enter the name of the axis to be cleared in the input box example: Enter (X1) press F3 "Relative coordinates Clear" and X1 coordinates will appear as 0
	4	**The relative coordinates are clear**	Press F4 "relative coordinates fully clear", then all axial displays of the relative coordinates display screen will be 0
	5	**Workpiece coordinate system**	"Workpiece coordinate system" is to set the external offset, G54,G55,G56 data position, please use with caution

4.10 Tool management

4.10.1 **Wear input**

Reference:	The machine will have a 901 alarm if the maximum wear input is +-3.0mm Input data to write the decimal point, if there is no decimal point, the system default is 0.001mm The screen of wear correction is the screen of the input tool wear

correction

The operation mode of wear input can be divided into "direct input" or "incremental input".

"Direct input" is to fill the value in the input box directly into the position to be changed

"Incremental input" is to fill the original value of the value + in the input box into the position to be modified

（图 1）



（图 2）



【 **Operation method** 】

1. Press the left ◀ or right ▶ button to return to the main page (Figure 1).

2. Press F3 刀具管理 to enter the tool compensation screen.

3. Press F1 磨耗 to enter the screen of wear setting, as shown ☜ by the arrow in the upper left corner.

4. Press F5 直接输入 or F6 增量输入 to select the calculation method (it turns to dark gray after selecting).

5. Press the keyboard ↑ ↓ to move the cursor to the axis to be compensated (cursor is displayed as white) and enter the data to be compensated in the input box for the position to be compensated.

6. Press the keyboard ENTER 回车 to finish.

4.10.2 **Shape input**

Reference:	The "Shape Input" Max of +-100.0mm will cause a 901 alarm if the machine exceeds the maximum input value Input data to write the decimal point, if there is no decimal point, the system default is 0.001mm The shape correction screen is the screen of the input tool shape correction Shape input operation mode is divided into "direct input", "incremental input", "measurement input". "Direct input" is to fill the value in the input box directly into the position to be changed "Incremental input" is to fill the original value of the value + in the input box into the position to be modified "Measurement input" is to calculate the current position and the input value after automatically filling in the modified position, usually used to match the knife



【 Operation method 】

1. Press left ◀ and right ▶ to return to the main page (Tool Management).

2. Press F3 刀具管理 to enter the tool compensation screen.

3. Press F1 形状 to enter the shape setting screen, as shown ☜ by the arrow in the upper left corner.

4. Press F5 直接输入 or F6 增量输入 ,F7 测量输入 to select the calculation method (it turns dark gray after selection).

5. Press the keyboard ↑ ↓ to move the cursor to the axis to be compensated (cursor is displayed as white) and enter the data to be compensated in the input box for the position to be compensated.

6. Press the keyboard ENTER 回车 to finish.

4.10.3 Setting the knife tip

Reference:	"Tip Settings" Max. +-100.0mm. Machine tools that exceed the maximum input value will display a 901 alarm Input data to write the decimal point, if there is no decimal point, the system default is 0.001mm The tool tip is divided into "tool tip radius, tool tip wear, tool tip direction" three specific can refer to the new generation of programming instructions G41,G42 tool tip radius compensation chapter Tip input operation mode is divided into "direct input, incremental input". "Direct input" is to fill the value in the input box directly into the position to be changed "Incremental input" is to fill the original value of the value + in the input box into the position to be modified



【 Operation method 】

7. Press left ◀ and right ▶ to return to the main page (Tool Management).

8. Press F3 刀具管理 to enter the tool compensation screen.

9. Press F3 刀尖 to enter the knife tip setting screen, as shown by the arrow in the upper left corner.

10. Press F5 直接输入 or F6 增量输入 to select the calculation method (which turns dark gray).

11. Press the keyboard [↑] [↓] to move the cursor to the axis to be compensated (cursor is displayed as white).

12. Enter the data to be compensated in the input box.

13. Press the keyboard [ENTER 回车] to finish.

4.10.4 **Tool length input**

⚠ WARNING

1. "Tool length input" is used by machine tool manufacturers to store tool location data. Non-professional maintenance personnel shall not modify it
2. Under normal circumstances, the tool length display screen is hidden to prevent the operator from misoperating and modifying the value, which may cause the deviation of the tool center position.
3. "Tool length input" Maximum +-600.0mm beyond the maximum input value of the machine will appear 901 alarm
4. To display the knife length screen, move the cursor to the position of T01 knife X1 in the "Shape input" mode, enter 52666, and then press the key [ENTER 回车] on the keyboard
5. The knife length screen display time can modify the value of parameter #3446

（图 1）



（图 2）



【 Operation method 】

1. Press left ◀ and right ▶ to return to the main page (Tool Management).

2. Press F3 [刀具管理] to enter the tool compensation screen.

3. Press F2 [形状] to enter the shape correction screen.

4. Press the keyboard [↑] [↓] to move the cursor to the position X1 of knife T01.

5. Enter 52666 key [ENTER 回车] knife long screen to display.

6. Press F4 [刀长] to enter the knife setting screen, as shown ☜ by the arrow in the upper left corner.

7. Press F5 [直接输入] or F6 [增量输入]. F7 [测量输入] selects the calculation method (it turns dark gray after selection).

8. Press the keyboard [↑] [↓] to move the cursor to the axis to be compensated (cursor is displayed in white), and enter the data to be compensated in the input box for the position to be compensated.

14. Press the keyboard [ENTER 回车] to finish.

4.10.5 **Tool life management**

Reference:	Tool Life Management can manage up to 50 tools The Tool Life Management page is displayed with parameter #3228 set to 1 The tool life count is valid only when "Tool Life" is enabled and "Empty Run" is off in "Function Settings" Tool life counts refer to the programming manual M97 using method counts When the tool life reaches, the machine will appear "10001 tool life reaches" alarm, and the machine will keep "suspended" state

（图 1）



(图 2)



Operation	No	Name	Function
寿命管理	1	Knife number	The number of the tool used in machining the part
	2	Knife store	Current sleeve number of the tool (standard is 0)
	3	group	The same group belongs to the same type of cutter. If the first knife in this group is locked or its life has reached its end, when T code is used to change knives, the first knife will be skipped and the second knife will be selected, and so on, it is recommended to set it to 0
	4	Knife number consulting	L - Locked status/U - Unlocked status ² Locked state, the tool cannot be used, T code will skip the tool change B -- Large diameter tool/N -- Standard tool ² Adjacent sleeves of large diameter tool must be empty (display only) T -- Use time/C -- use times ² Determines the current life, maximum service life, life warning display and use unit as times or time R -- The value is valid / -- The value is invalid ² Whether the tool uses the tool life management function

Operation	No	Name	Function
	5	Current life	Current tool usage data recording
	6	Maximum life	Maximum tool life The machine will alarm when the current life is greater than or equal to the maximum life
	7	Life prediction	A notice will be issued when the tool life is longer than predicted
	8	Tool condition	I (0) Unmanaged: The set value is invalid I (1) Unused: Tool life is 0 I (2) available: 0 < Tool life < Life prediction I (3) End notice: Life notice < Tool life < Maximum life I (4) End of life: maximum life < Tool life I (5) Tool damage

【 Operation method 】

1. Press left ◀ and right ▶ to return to the main page (Tool Management).

2. Press F8 寿命管理 to enter the tool life management screen (Figure 1).

3. Press the keyboard ↑ ↓ to move the cursor to the desired position.。



4. The following figure describes the function of the area covered by the yellow cursor on the right of the input box

5. When the cursor is moved to the desired position, enter the desired value in the text box.

6.Press the keyboard ENTER 回车 to finish.

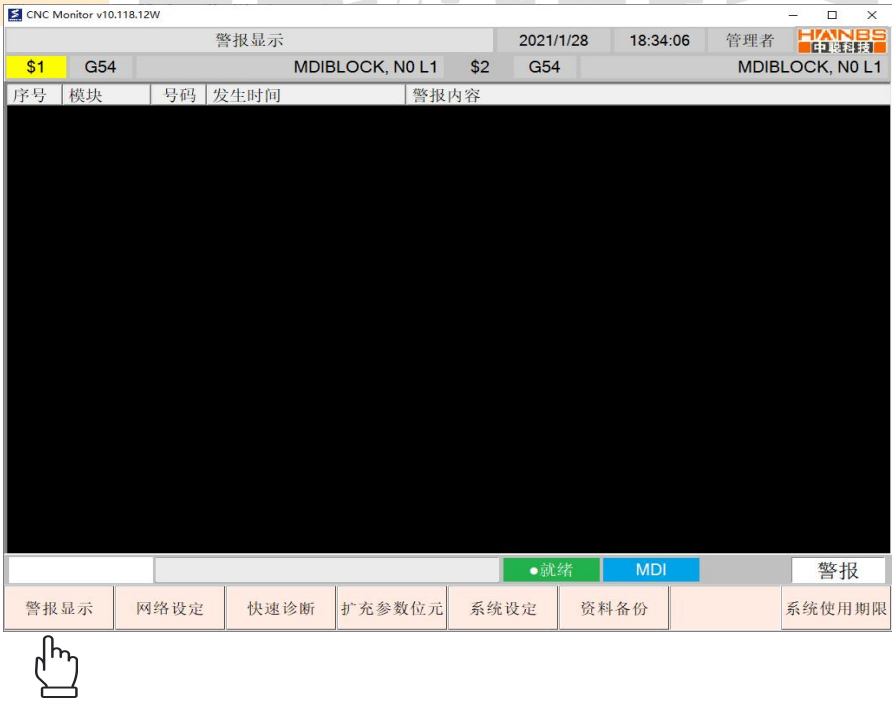
4.11 Maintaining

4.11.1 **Alarm display**

Reference:	When the alarm occurs due to mechanical failure, the content information will be displayed on the screen. After a fault occurs, first investigate whether there is a display message. Then rectify the fault. When starting after handling the cause of lifting the alarm, please fully confirm the relevant position of each part of the machine before starting

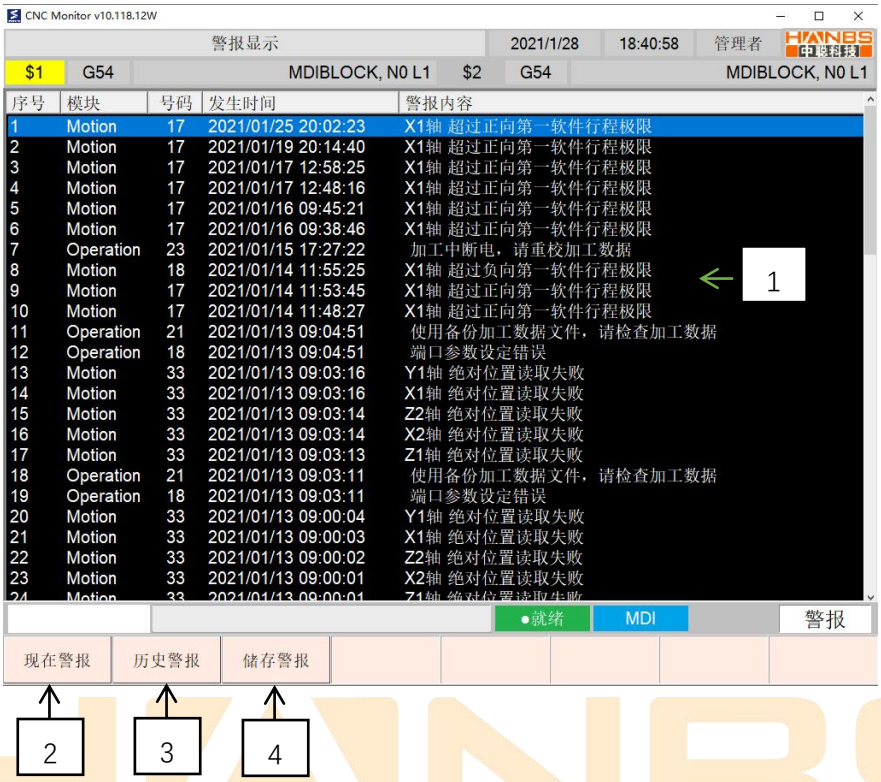
（图 1）


CNC Monitor v10.118.12W
机台坐标
2021/1/28
18:24:49
管理者
$1
G54
MDIBLOCK, N0 L1
$2
G54
MDIBLOCK, N0 L1
机械座标
X1 -122.010
Y1 -25.915
Z1 0.000
C1 0.000
机械座标
X2 0.000
Z2 0.000
C2 0.000
Y2 0.000
F 1.0 mm/min 100 %
0.0 mm/min (实际)
S 1000 RPM 100 %
0 RPM (实际)
就绪
MDI
警报
机台坐标
程序编辑
刀具管理
加工监控
维护
用户参数
功能设置
辅助面板


CNC Monitor v10.118.12W
警报显示
2021/1/28
18:34:06
管理者
$1
G54
MDIBLOCK, N0 L1
$2
G54
MDIBLOCK, N0 L1
序号
模块
号码
发生时间
警报内容
就绪
MDI
警报
警报显示
网络设定
快速诊断
扩充参数位元
系统设定
资料备份
系统使用期限

（图二）

（图 3）

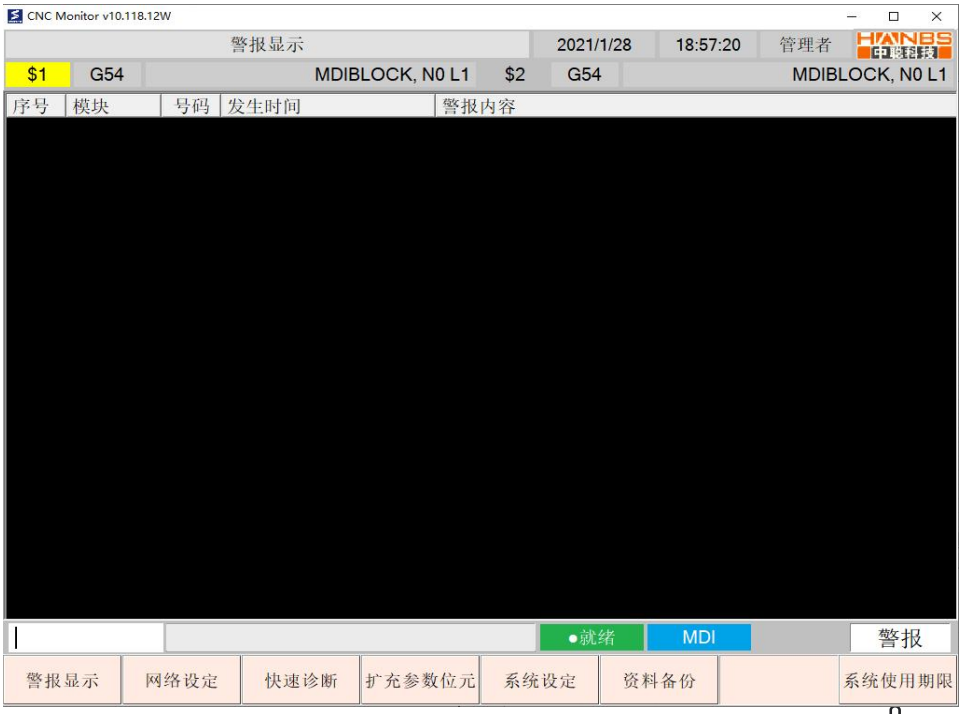


Operation	No	Name	Function
警报显示	1	**Alarm display area**	Alarm display area, including alarm details
	2	**Now alert**	An alert that has occurred in the current system
	3	**Historical alert**	An alarm that has occurred in the system
	4	**Store alarm**	Alarms that have occurred can be stored in a file

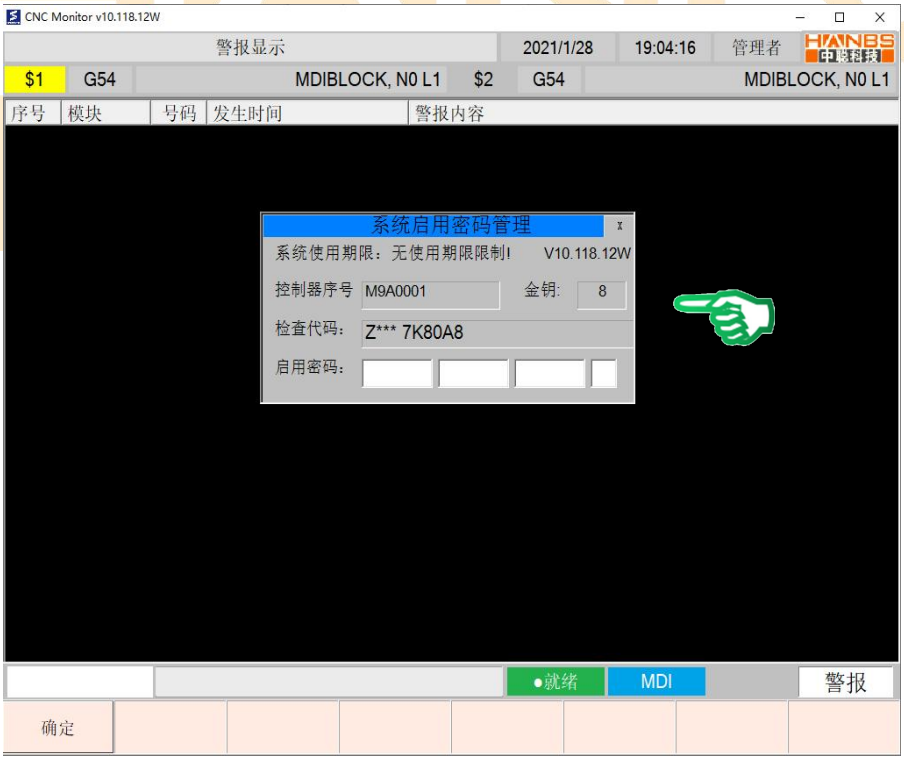
4.11.2 System life

Reference:	When the password expires, the machine will prompt you to enter the correct password provided by the machine tool factory. When the password is about to expire, the machine will automatically remind the remaining service life

（图 1）



（图 2）



【 Operation method 】

1. Press the left ◄ or right ► button to return to the main page.

2. Press F5 [维护] to go to the maintenance page.

3. Press F8 [系统使用期限] to enter the password setting screen.

4. Press the key corresponding to the keyboard according to the password provided by the machine tool manufacturer.

5. Press the keyboard [ENTER 回车] or F1 [确定] key after the input is complete.

6. If a message is displayed indicating that the password is incorrect, enter the password again by following the preceding steps.

4.12 User parameter

⚠WARNING	Non-professional personnel are not allowed to modify the data in User Parameters If necessary, it can be modified under the guidance of the manufacturer's after-sales staff Before modifying the parameters, ensure that the machine is stopped in a safe position Power off and restart after modifying parameters Please refer to New Generation Parameter Specification for specific function of parameters.



【 Operation method 】

1. Press the left ◀ or right ▶ button to return to the main page.
2. Press F6 用户参数 to access the user parameters page.
3. Press the keyboard ↑ ↓ to move the cursor to the position where you want to modify the parameter.
4. In the input box, enter the value to be set.
5. Press the keyboard ENTER 回车 or F1 确定 key after the input is complete.

4.13 Function settings

4.13.1 **Switching on/off with or without guide bushing**

Reference:

Guide sleeve mode is divided into two types: "no guide sleeve" and "with guide sleeve"
The maximum length of processing without guide sleeve is 60.0mm
The maximum length with guide sleeve is 195.0mm
Ensure safe position of tool handling when switching guide bush
The guide sleeve can only be switched in manual mode; otherwise, the function keys are grey

（图 1）

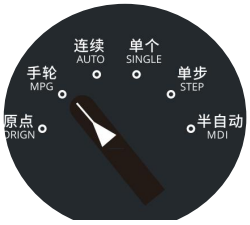

CNC Monitor v10.118.12W
机台坐标
2021/1/28
19:50:53
管理者
$1 G54 MDIBLOCK, N0 L1
$2 G54 MDIBLOCK, N0 L1
机械座标
X1 -122.010
Y1 -25.915
Z1 0.000
C1 0.000
机械座标
X2 0.000
Z2 0.000
C2 0.000
Y2 0.000
F 1.0 mm/min 100 %
0.0 mm/min (实际)
S 1000 RPM 100 %
0 RPM (实际)
就绪
MDI
警报
机台坐标
程序编辑
刀具管理
加工监控
维护
用户参数
功能设置
辅助面板

（图 2）


CNC Monitor v10.118.12W
功能设置
2021/1/28
21:24:42
管理者
$1 G54 S9B07--THX, N0 L1
$2 G54 S9B07--THX, N0 L1
功能设置
导套模式 无导套模式
刀具寿命启用/禁用 启用
自动送料机 不使用
备用开关
实速空运行 禁用
侧面动力维护计时（小时） 0/2000
背面动力维护计时（小时） 0/2000
* 请于手轮模式下对本页功能进行操作并确认安全！
v2.09
就绪
手轮
警报
导套模式
换夹模式
刀具寿命
自动送料机
备用开关
空运行
维护完成

【 Operation method 】

1. Ensure that the machine is in a safe position.

2. Twist the mode switch 

to the handwheel position.

3. Press the left ◀ and right ▶ keys to find the main page (Figure 1).

4. Press F7 功能设置 to enter the function setting page (Figure 2).

5. Press F1 导套模式 key to switch in the guide mode prompt box on the screen (with/without guide) to switch with/without guide completed.

4.13.2 **Clip change mode switch on/off**

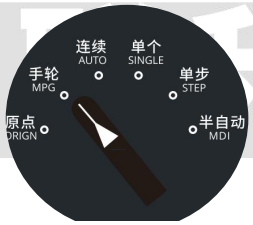
Reference:	Make sure the machine is in a safe position when replacing the chuck Make sure to switch back to the chuck mode after replacing the chuck Only in manual mode can you switch the clip change mode; otherwise, the function button is grey

（图 2）



【 Operation method 】

1. Ensure that the machine is in a safe position.



2. Twist the mode switch to the handwheel position.

3. Press the left ◀ and right ▶ keys to find the main page (Figure 1).

4. Press F7 [功能设置] to enter the function setting page (Figure 2).

5. Press F2 [换夹模式] to switch in the guide mode prompt box on the screen (with/without guide change clip mode)

4.13.3 **Tool life on/off**

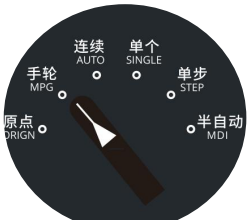
Reference:	When executing the program M97 tool count, only the tool life switch "enabled" is effective, otherwise the tool life management page count function is invalid For details about tool life management, see the section on Tool Life

（图 2）



【 Operation method 】

1. Ensure that the machine is in a safe position.



2. Twist the mode switch to the handwheel position.

3. Press the left ◀ and right ▶ keys to find the main page (Figure 1).

4. Press F7 [功能设置] to enter the function setting page (Figure 2).

5. Press F3 [刀具寿命] key to switch in the tool life prompt box on the screen (Disable/Enable)

6. The switch with or without guide sleeves is complete

4.13.4 **Use on/off for feeder**

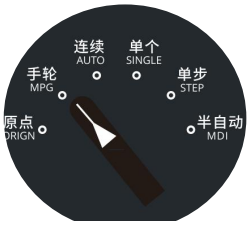
Reference:	"Feeder switch" open state if the feeder is not in the "automatic state", the machine tool will appear alarm and prohibit the machine from starting The "feeder switch" off state, such as the feeder is not in the "automatic state" under the machine can also execute the processing program, may cause the length of the processing parts is not stable or knife collision and other adverse consequences Only in manual mode can the feeder mode be switched, otherwise the function keys will be grey

（图 2）



【 Operation method 】

1. Ensure that the machine is in a safe position.



2. Twist the mode switch to the handwheel position.
3. Press the left ◀ and right ▶ keys to find the main page (Figure 1).
4. Press F7 功能设置 to enter the function setting page (Figure 2).
5. Press F4 自动送料机 key to switch in the automatic feeder prompt box on the screen (use/not use)
6. The switch between using and not using the feeder is completed.

4.13.5 **Stand by on/off**

⚠ PAY ATTENTION
➢ Spare switches are left to developers as backups. ➢ If the operator wants to use this switch, please contact the machine tool factory technical personnel, and use under the guidance of professional personnel.



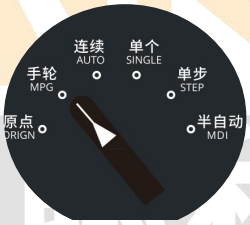
4.13.6 **Run empty on/off**

Reference:	Please reduce the fast forward rate when empty running is valid No valid/invalid switch for empty running during machine operation Tool life management is invalid when empty operation is effective Only in manual mode can the empty operation mode be switched; otherwise, the function keys will be grey



【 Operation method 】

1. Ensure that the machine is in a safe position.



2. Twist the mode switch to the handwheel position.
3. Press the left ◀ and right ▶ keys to find the main page (Figure 1).
4. Press F7 [功能设置] to enter the function setting page (Figure 2).
5. Press F6 [空运行] to switch in the prompt box (Disable/Enable)

4.13.7 **Power head maintenance timing**

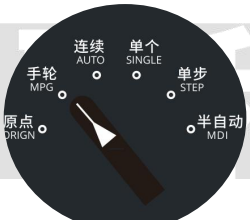
⚠PAY ATTENTION

- After the service life of the power head reaches 2000 hours, an alarm will appear on the machine tool to prompt technical personnel to repair the power head. For details, refer to the machine tool maintenance manual
- If the time is not maintained may damage the power head gear and other accessories



【 Operation method 】

1. 1. Maintain the side milling and back shaft power head. Refer to the maintenance manual provided by the machine tool manufacturer for specific operations.



2. Twist the mode switch to the handwheel position.

3. Press the left ◀ and right ▶ keys to find the main page .

4. Press F7 功能设置 to enter the function setting page .

5. Press F8 维护完成 to cancel the alarm

4.13.8 Auxiliary panel

⚠ PAY ATTENTION

- Please ensure that all parts of the mechanical rotation parts are locked when using.
- Ensure operation safety door is closed.
- Ensure that the tool and fixture do not interfere.
- Each function is effective only when the manual device is used, otherwise the key will be displayed grey and cannot be used.

（图 1）



（图 2）

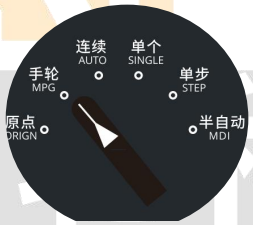


Operaton	No	Name	Function
辅助面板	1	**Axial load monitoring**	Axial load monitoring, convenient for operators to understand the axial running state of the machine
	2	**Spindle speed monitoring**	Speed monitoring for manual spindle rotation
	3	**Axial load**	Press F1 轴向负载 to switch to the axial load monitoring screen
	4	**Positive spindle rotation**	Press the F3 主轴正转 spindle clockwise in "Manual mode" and continuously press the F3 spindle to stop the rotation
	5	**Spindle reversal**	Press F4 主轴反转 spindle counterclockwise in "Manual mode" and continuously press F4 spindle to stop rotation
	6	**Positive**	Press F5 副轴正转 subspindle to rotate clockwise in

Operaton	No	Name	Function
		rotation of the auxiliary axis	"manual mode" and continuously press F5 subspindle to stop rotating
	7	**Counteraxis inversion**	In "manual mode", press F6 [副轴反转] subspindle counterclockwise to rotate, and continuously press F6 subspindle to stop rotating
	8	**Positive side power turn**	Press the F7 [侧动力正转] side power head to rotate clockwise in "manual mode", and continuously press the F7 side power head to stop the rotation
	9	**Side dynamic reversal**	In "Manual mode", press the F8 [侧动力反转] side power head to rotate counterclockwise and continuously press the F8 side power head to stop the rotation

【 Operation method 】

1. Ensure that the machinery, tools and fixtures are in a safe position and locked.



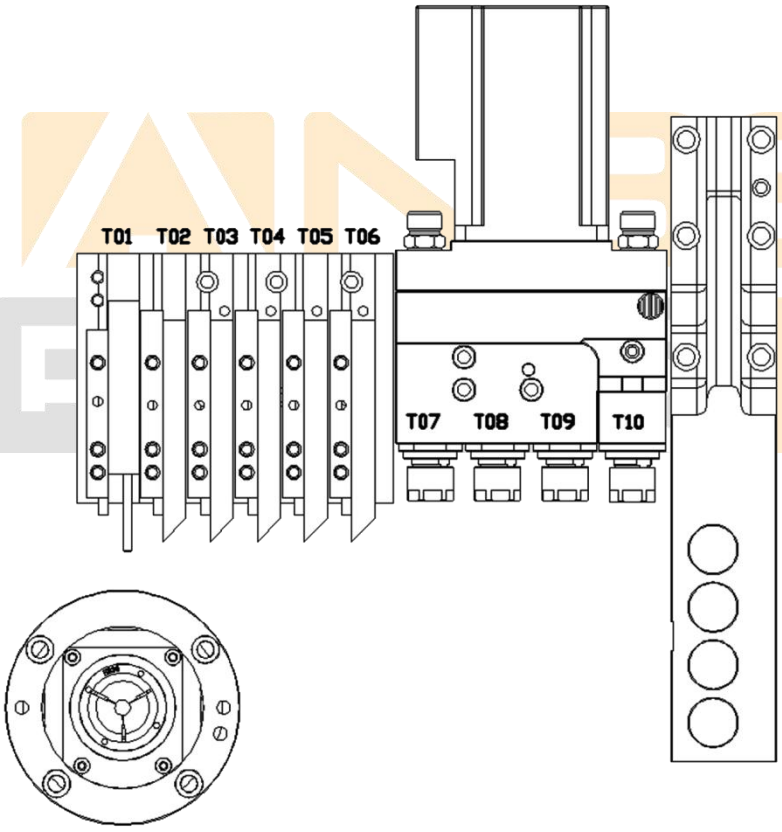
2. Twist the mode switch to the handwheel position.
3. Press the left ◀ and right ▶ keys to find the main page (Figure 1).
4. Press F8 [辅助面板] to go to the auxiliary panel page (Figure 2).
5. Press the corresponding key (F1-F8) as required by the operator.

Chapter Ⅴ Tool installation

5.1 Tool row installation

⚠PAY ATTENTION

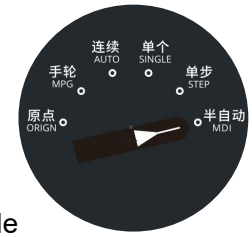
- In MDI mode, please choose the appropriate magnification to avoid knife collision.
- Note When operating in manual mode, note the moving direction. Misoperation will cause damage to tools and machines.
- After tool installation, please return the material to the guide sleeve to prevent collision.
- In order to ensure safety, please open the hand wheel simulation.



Row tool number T01-T06, T01 is usually used to install the cutting detection device, T02-T06 is used to install the turning tool.

The following takes the diameter of T03 tool Ø10.0mm as an example. The installation sequence is as follows:

【 Operation method 】

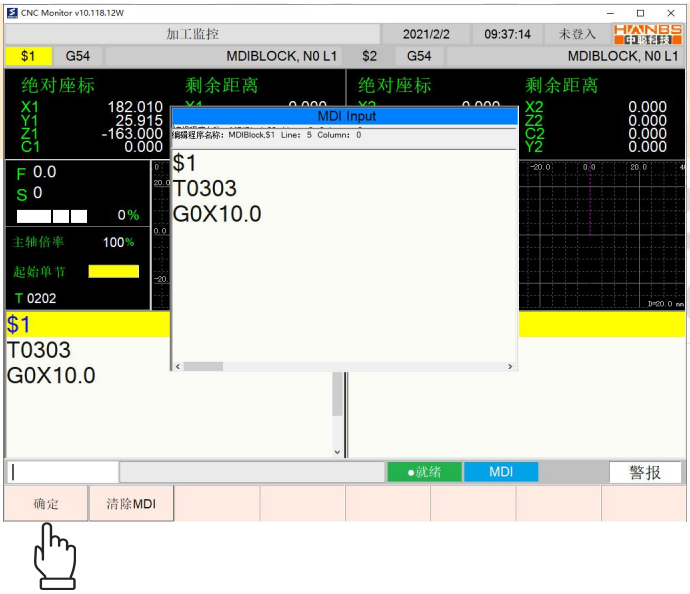
1. Check that the tool and material are in a safe position and will not collide.
2. Pull the material out of the guide sleeve.
3. Switch mode to MDI semi-automatic mode 

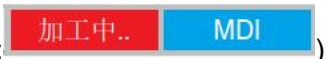
.
4. Press the Channel 1 key on the operation panel (select System 1)

.
5. The hand wheel on the operation panel is simulated lit

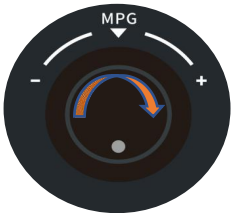
.
6. Press the on-screen MDI button

.
7. Enter the tool number and material diameter in the screen input box and press F1 to confirm.



8. Ensure the safe position of the machine, press the start button  to execute the MDI input program (Note: If the program start state is "processing", as shown in Figure:

)

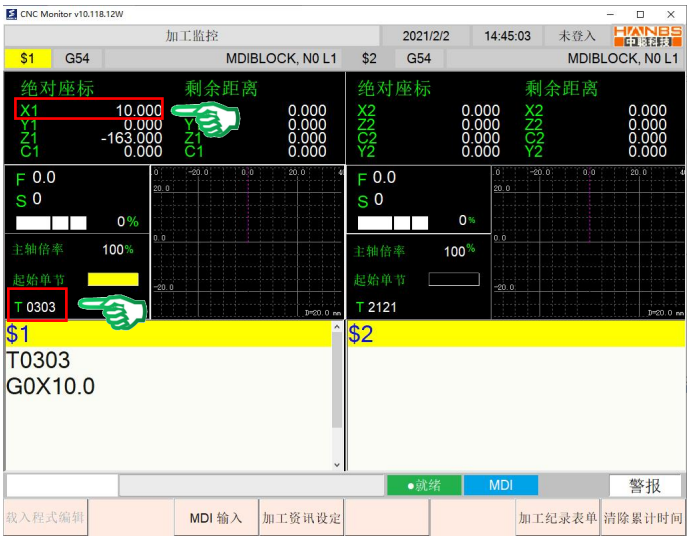


9. Turn the hand wheel clockwise until the above process is completed and the warning light shows yellow.

10. MDI program processing is completed (Note: After program processing is completed, the state is displayed as "Ready". 

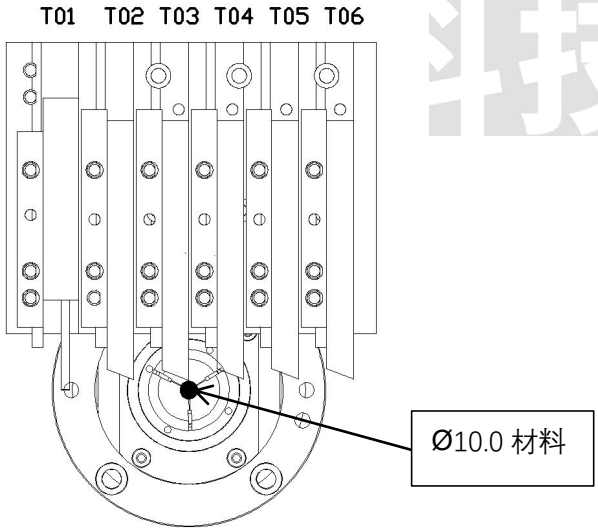
)

11. Check whether the program coordinates are the diameter of the material, and whether the tool number is correct, as shown below.



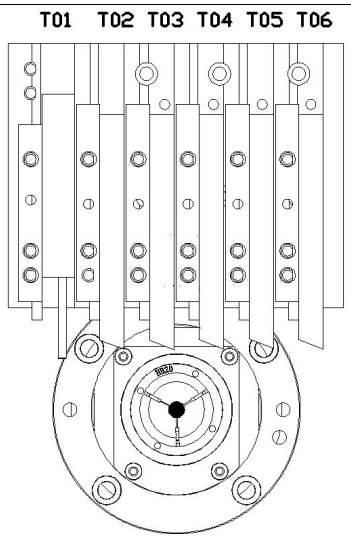
12. Rotate the hand wheel clockwise to remove the material from the guide sleeve.

13. Install the tool to the T03 tool position so that the tool tip can contact the material and gently tighten the fixing bolt as shown below.



14. Press the operating panel key 

and rotate the "hand wheel" clockwise to leave the tool tip from the workpiece.

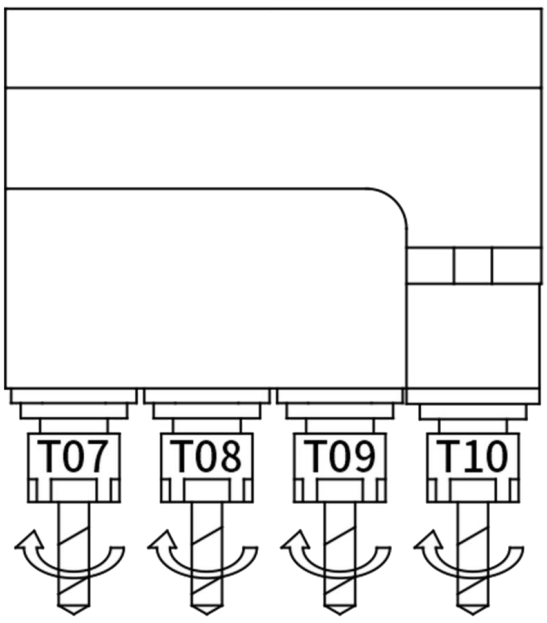


15. Tighten turning tool bolt, tool installation is complete.

5.2 Cutter installation of milling unit

⚠ PAY ATTENTION

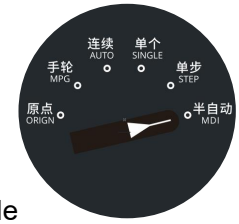
- Before installing the milling cutter, please clean the wear inside cutter (can be ignored in special cases).
- Before installing the tool, please make sure that the material is outside the guide sleeve (if it is outside, please return the material to the guide sleeve).
- Note When operating in manual mode, note the moving direction. Misoperation will cause damage to tools and machines.
- After tool installation, please return the material to the guide sleeve to prevent collision.
- In order to ensure safety, please open the hand wheel simulation.



The cutter number of the side milling unit is T07-T10, where T10 can be selected from different tool combinations, and the chuck size is ER16.

Take T07 tool diameter 10.0 as an example. The installation sequence is as follows:

【 Operation method 】

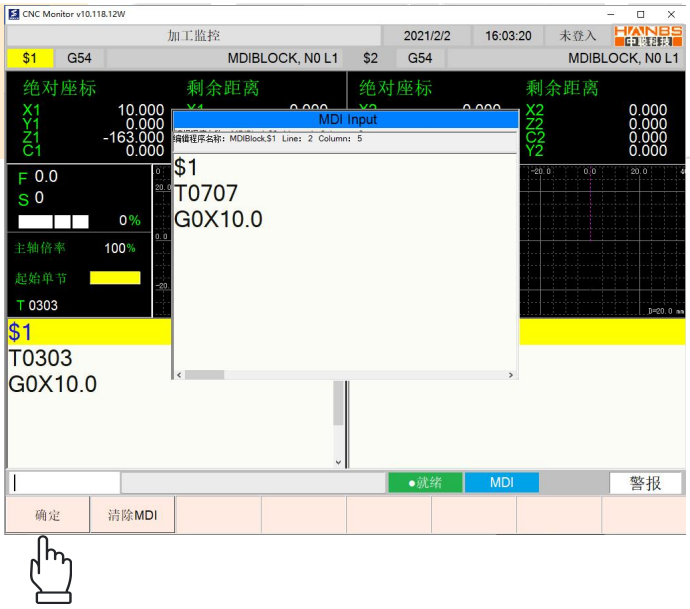
1. Ensure that the machine is stopped.
2. Pull the material out of the guide sleeve.
3. Switch mode to MDI semi-automatic mode 

.
4. Press the Channel 1 key on the operation panel (select System 1)

.
5. The hand wheel on the operation panel is simulated lit

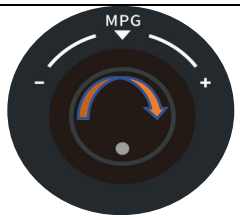
.
6. Press the on-screen MDI button

.
7. Enter the tool number and material diameter in the screen input box and press F1 to confirm.



8. To ensure the safe position of the machine, press the start button  to execute the MDI input program (Note: If the program start state is "processing", as shown in the figure:

)

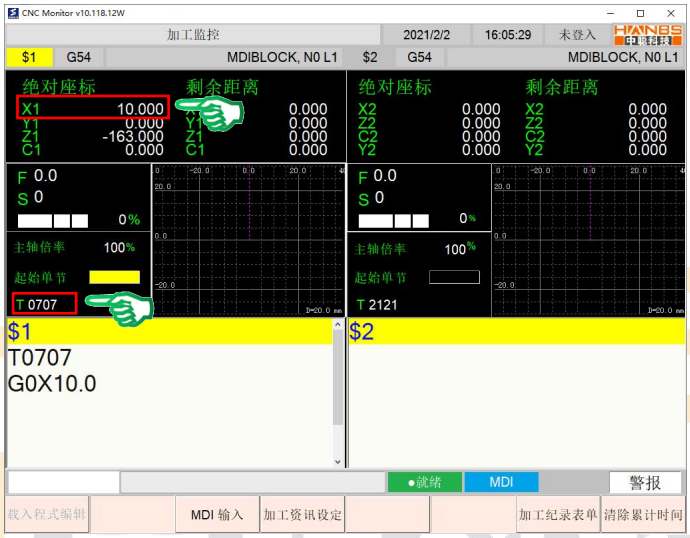


9. urn the hand wheel clockwise until the above process is completed and the warning light shows yellow.

10. MDI program processing is completed (Note: After program processing is completed, the state is displayed as "Ready". 

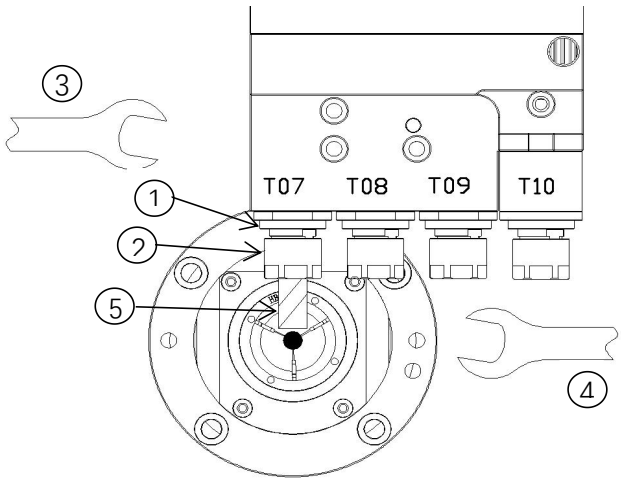
)

11. Check whether the program coordinates are the diameter of the material, and whether the tool number is correct, as shown below.



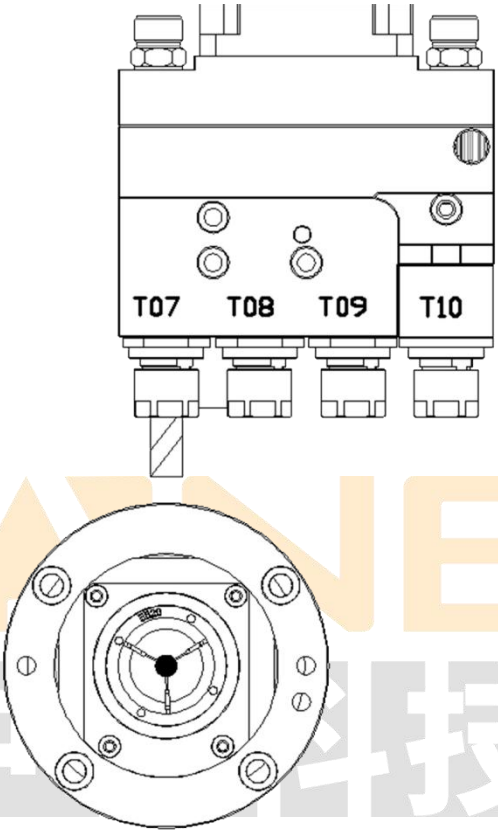
12. Turn the hand wheel clockwise to remove the material from the guide sleeve.

13. Loosen the cap ② and install the cutting tool ⑤ into the sleeve. The protrusion from cap to end cutting tool can be determined according to machining depth and chip state.



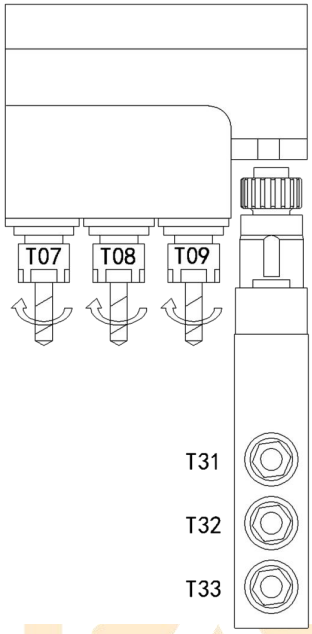
14. Use the rotary stop special plate ③ and cap special plate ④ to tighten the cap.

15. Press the operating panel key [X] and rotate the "hand wheel" clockwise to remove the milling cutter from the workpiece.



16. Tighten turning tool nuts, tool installation is complete.

5.3 Extension of the end milling unit

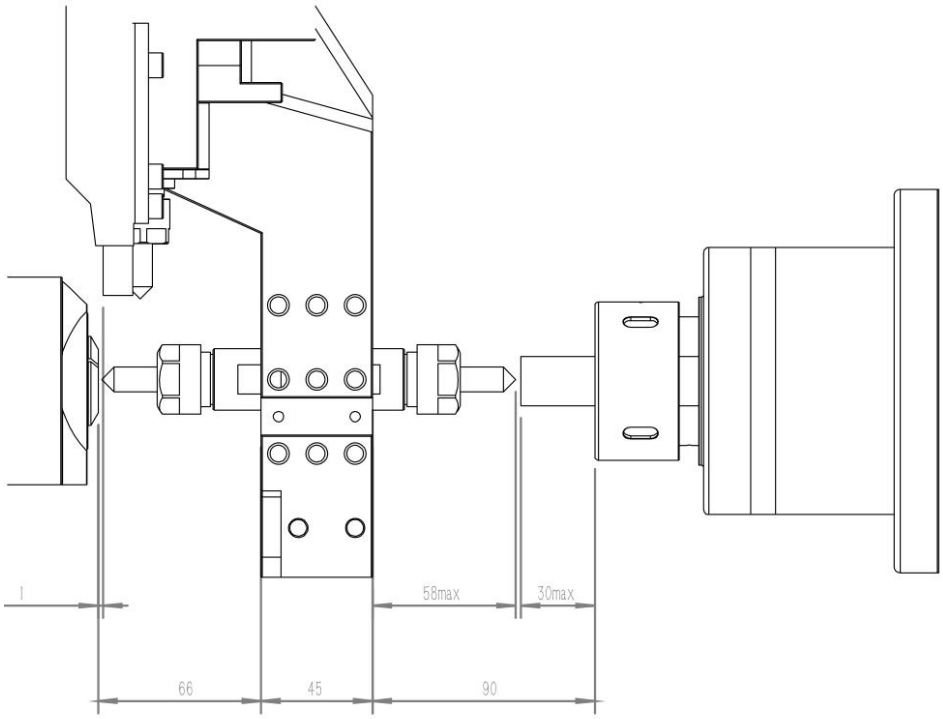


Cutter number of face milling unit T31-T33 Chuck size ER11(two-way cutter loading). Refer to side milling unit for the specific tool installation method.

5.4 Installing a fixed drilling tool

⚠PAY ATTENTION

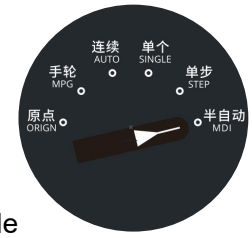
- Before installing the drill, remove the wear inside tool (can be ignored in special cases).
- Before installing the tool, please make sure that the material is inside the guide sleeve (if it is outside, please return the material to the guide sleeve).
- Note When operating in manual mode, note the moving direction. Misoperation will cause damage to tools and machines.
- After tool installation, please return the material to the guide sleeve to prevent collision.
- In order to ensure safety, please open the hand wheel simulation.



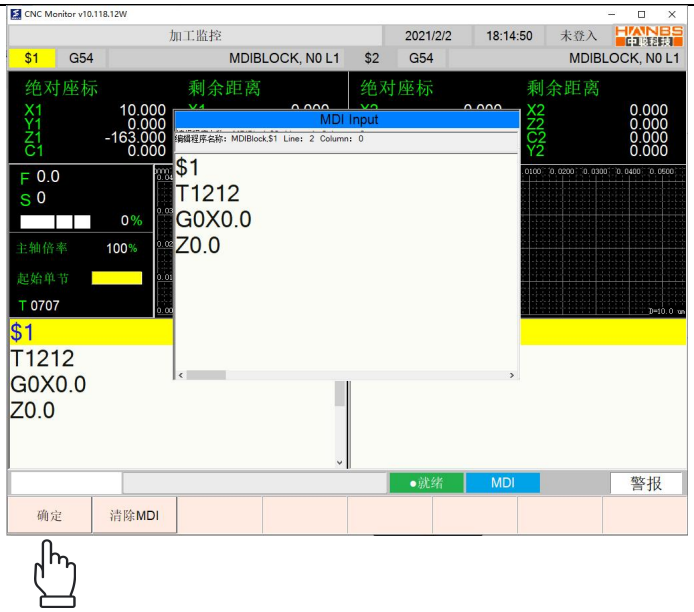
Explanation: The fixed drilling tool is T11-T14 shared by main shaft and auxiliary shaft. It can be used with single or double end extension rod according to actual needs. The tool installation sequence is as follows (T12) :

【 Operation method 】

1. Make sure there is material on the machine.
2. After the execution of G299,G300,G150 (workpiece length setting is completed).
3. Verify that the machine is stopped.
4. Confirm material is returned to guide sleeve (safe position).
5. Put the tool into the drilling seat, you can move back about 5 mm to prevent interference.
6. Turn on Handwheel mode  (Enable Handwheel mode).

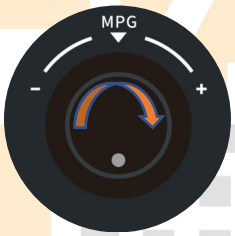


7. Switch mode to MDI semi-automatic mode .
8. Press the on-screen MDI button MDI 输入.
9. Enter the cutter number and Z 0.0 position in the screen input box and press F1 to confirm.



10. To ensure the safe position of the machine, press the start button  to execute the MDI input program (Note: If the program start state is "processing", as shown in the figure: 

)

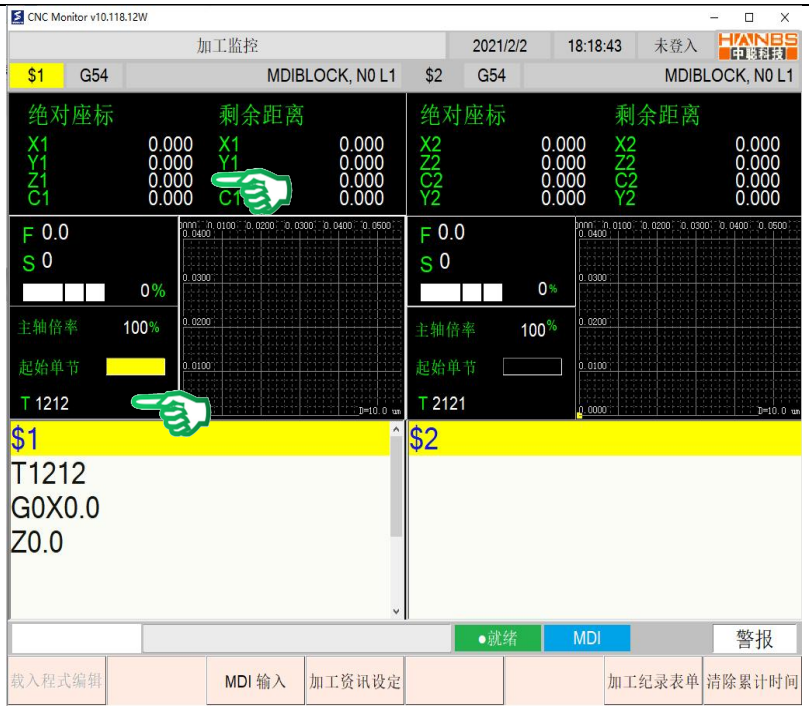
11. 

Turn the hand wheel clockwise until the above process is completed and the warning light shows yellow.

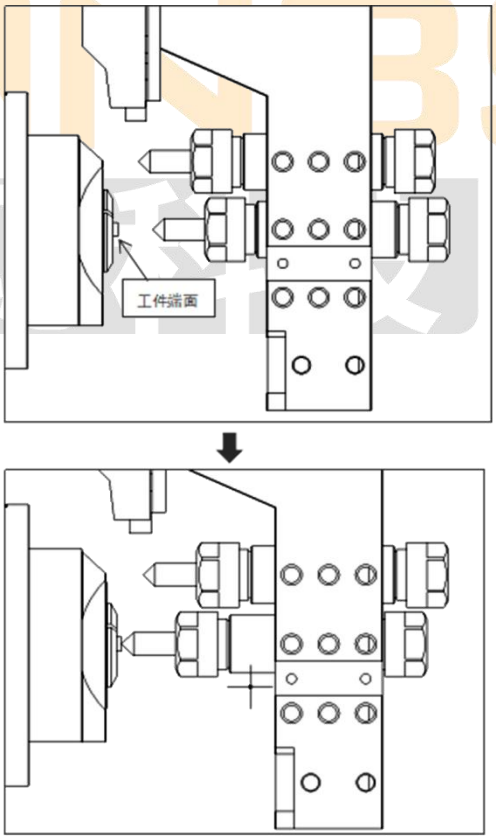
12. MDI program processing is completed (Note: After program processing is completed, the state is displayed as "Ready". 

)

13. Check the program coordinates Z0.0 and the knife number as shown below.

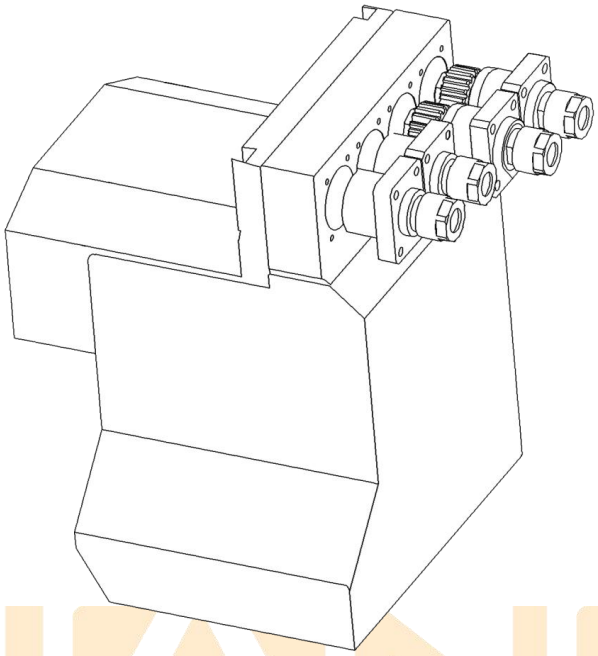


14. Make the tip of the knife touch the end of the material and lock the screw.



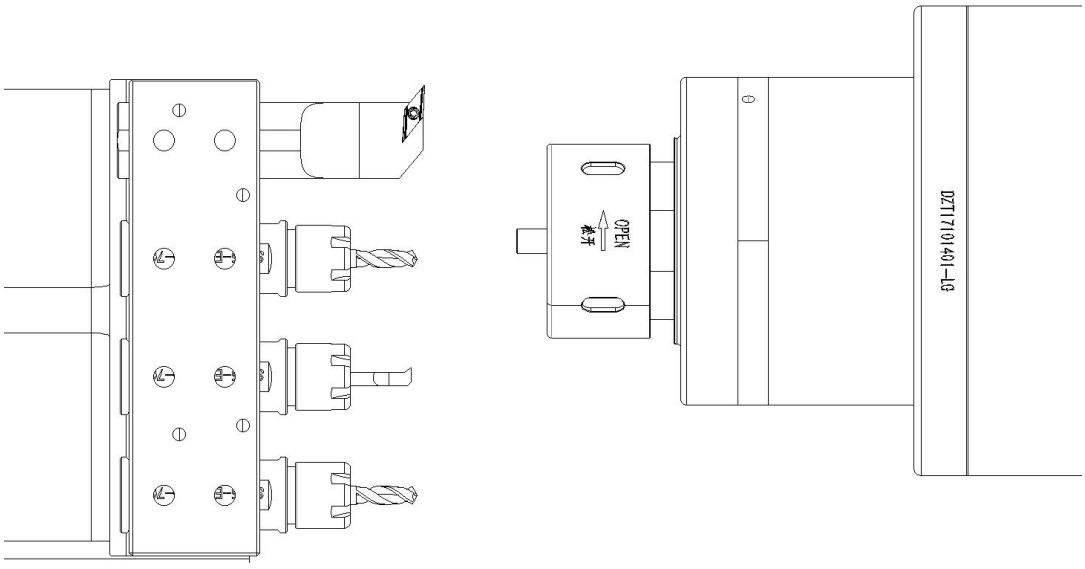
15. Turn the "hand wheel" counterclockwise to return the material to the guide sleeve.

5.5 Switch the fixed drilling tool



Explanation: The tool number of the back end milling unit is T21-T24 and the size of the collet is ER16. The four tools can be converted to each other. The fixed tool and movable tool can be matched, and the fixed tool and movable tool can be mixed and matched, and the back side drilling unit can be extended. Refer to side milling unit for the specific tool installation method.

5.6 Installing a rear tool



5.6.1 Install the "Measuring X2" axis of T21 turning tool

【 Operation method 】

1. Confirm that the workpiece of the auxiliary shaft is normally connected from the spindle end.

2. Make sure the machine is stopped.

3. Install the tool into the extension bar to the required length and tighten the screws (see picture).

4. Switch mode to "hand wheel" mode.



5. Press the "Auxiliary Panel" button on the screen 辅助面板.

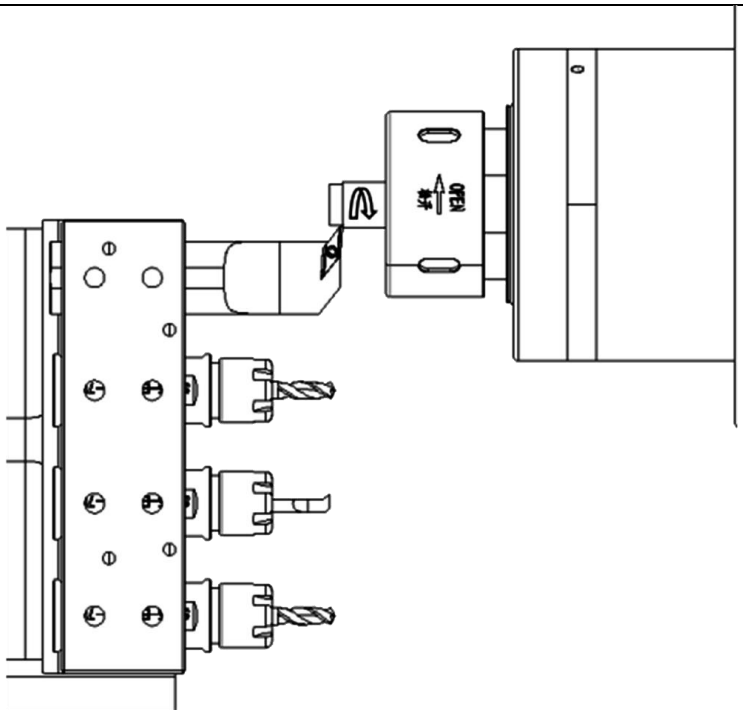
Reference: Face tool holder tool number T21-T24, usually can be installed with 4 fixed tools, or optional rotary tools.

For details about how to install rotary tools, see 3.9.2 Installing Milling Unit Tools.

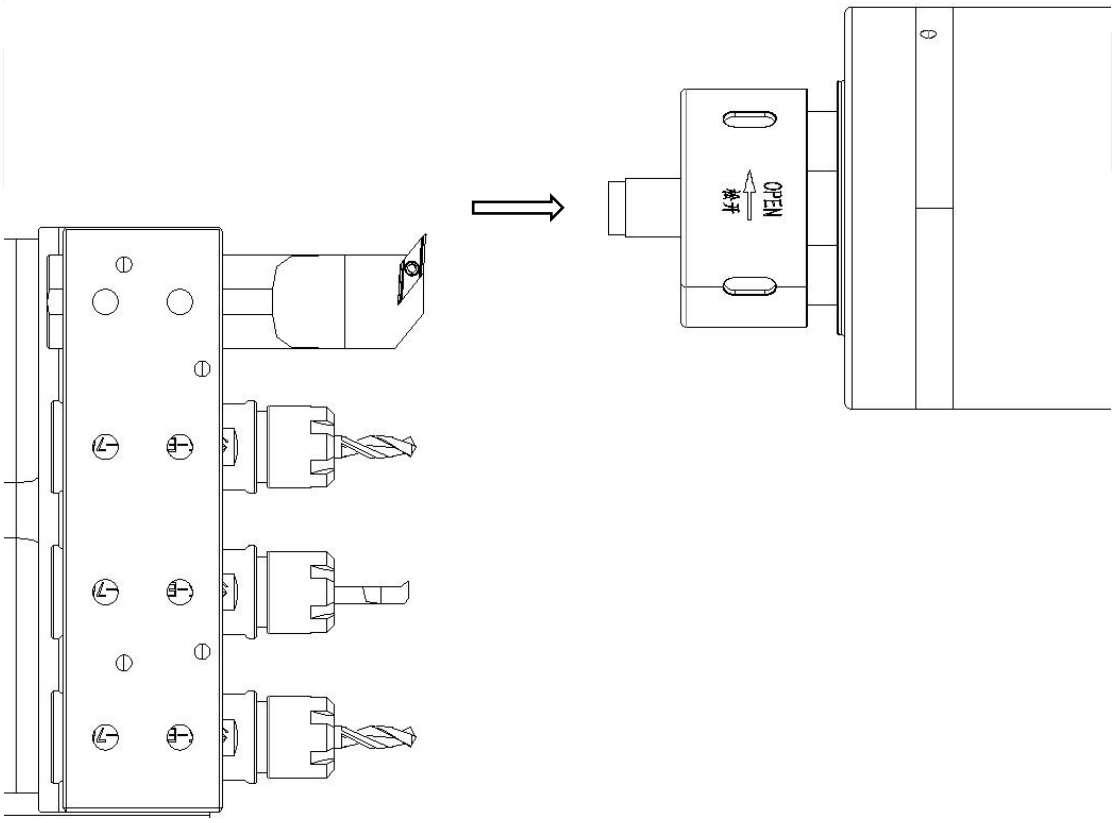
For details about how to install a fixed Drilling tool, see 3.9.3 Installing a Fixed Drilling Tool.



6. Press the button "counteraxis reversal" to rotate counteraxis counterclockwise as shown in the figure above.



7. Press the X and Z keys on the operation panel and rotate the auxiliary shaft in the direction of "hand wheel +-" to turn a section of the workpiece surface (as shown in the figure above).



8. Press the Z button on the operation panel, rotate the countershaft clockwise with the "hand wheel" to move the workpiece away from the turning position (as shown in the

figure above).

9. Press the Reset button on the operation panel. 



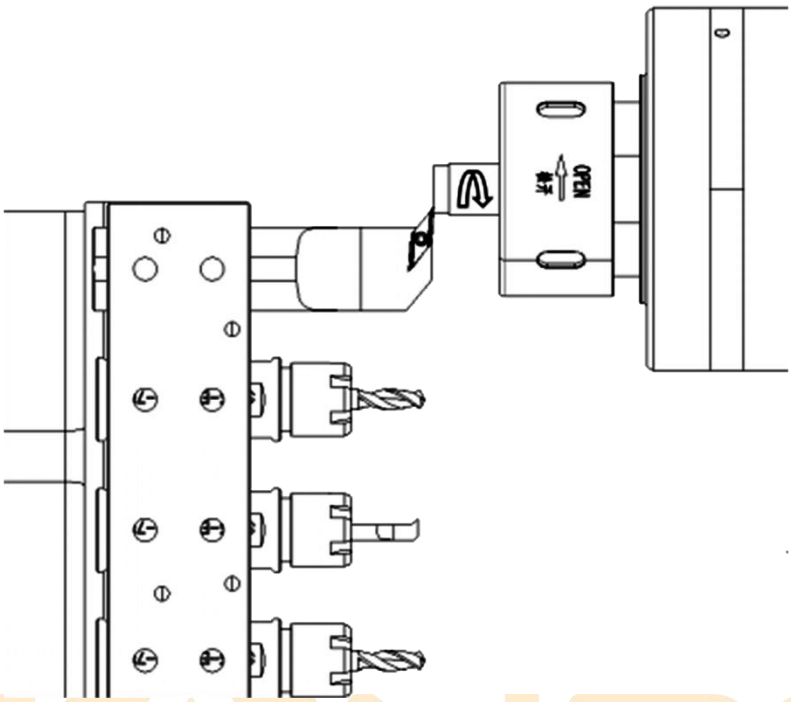
10. Measure the part of the workpiece being cut with caliper, input the value into "Tool Management, Shape", press the "Measure" key, the shape measurement of X2 axis is completed (as shown in the figure above).

5.6.2 Install "Measuring Z2" shaft of T21 turning tool

【 Operation method 】

1. Press the button "Counteraxis reversal" counteraxis rotation.

2. Press X and Z keys on the operation panel and rotate the auxiliary axis in the direction of "handwheel +-" to touch the end face of the workpiece (as shown in the figure).



3. Press the key to manage the tool "Shape", enter 0.0 and press the "Measurement" key. The shape measurement of Z2 axis is completed (as shown in the figure).



4. Press the Z button on the operation panel, rotate the countershaft clockwise with the "hand wheel" to make the machine tool in a safe position to complete the tool setting.

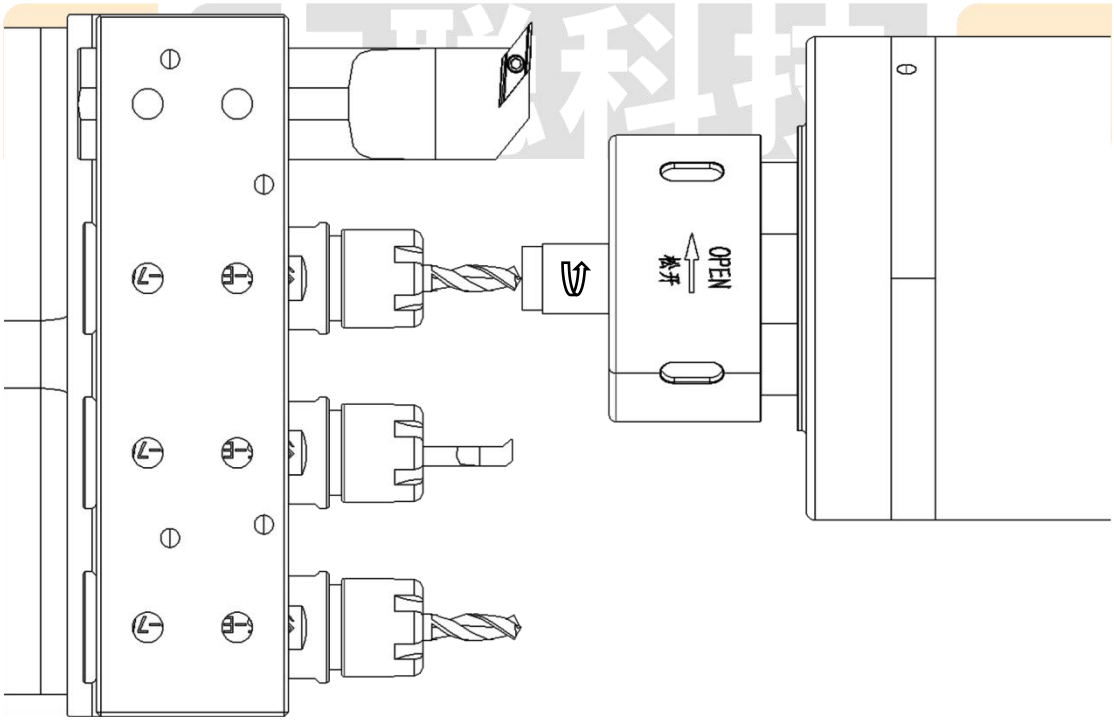
5.6.3 Install the "Measure Z2" shaft of the T22 twist drill

【 Operation method 】

1. 1. Press the button "positive rotation of the auxiliary axis" to rotate the auxiliary axis clockwise (as shown in the figure).



2. Press X and Z keys on the operation panel and rotate the auxiliary axis in the direction of "handwheel +-" to touch the end face of the workpiece (as shown in the figure).



3. Press the key to manage the tool "Shape", enter 0.0 and press the "Measurement"

key. The shape measurement of Z2 axis is completed (as shown in the figure).



4. Press the Z button on the operation panel, rotate the countershaft clockwise with the "hand wheel" to make the machine tool in a safe position to complete the tool setting.

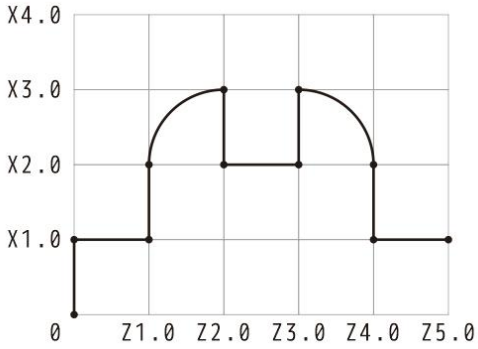
⚠PAY ATTENTION

- Before installing the drill, remove the wear inside tool (can be ignored in special cases).
- Make sure the back shaft is holding the workpiece before installing the tool.
- Note When operating in manual mode, note the moving direction. Misoperation will cause damage to tools and machines.
- After tool installation, please return the back shaft to the safe position.
- In order to ensure safety, please open the hand wheel simulation.

Chapter Ⅵ Programming instructions

6.1 NC program

A CNC program is usually a set of instructions that make a machine run. Each instruction consists of English letters and numbers. For example, if you want the machine to run in the following path, you may need to use the following instructions:



G01……Straight line movement

G02……Arc movement

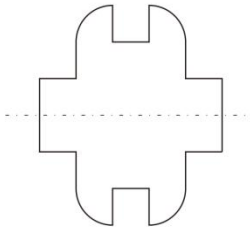
M08……Switch on

M09……Switch off

The complete program code is as follows:

```
0   M08;              (Switch on)
1   G01 X1.0 Z0;      (Move straight to X1.0, Z0)
2   G01 X1.0 Z1.0;    (Move straight to position X1.0, Z1.0)
3   G01 X2.0 Z1.0;    (Move straight to position X2.0, Z1.0)
4   G02 X3.0 Z2.0;    (Arc moved to X3.0, Z2.0 position)
5   G01 X2.0 Z2.0;    (Move straight to X2.0, Z2.0 position)
6   G01 X2.0 Z3.0;    (Move straight to X2.0, Z3.0 position)
7   G01 X3.0 Z3.0;    (Move straight to X3.0, Z3.0 position)
8   G02 X2.0 Z4.0;    (Arc moved to X2.0, Z4.0 position)
9   G01 X1.0 Z4.0;    (Move straight to position X1.0, Z4.0)
10  M09;              (Switch off)
```

If the above procedure is performed on a lathe, the product shown below will be produced.



6.2 Programming of numerical control programs

The machine works according to the stored machining program. In actual processing, a variety of processing factors need to be considered. The specific program writing sequence is as follows:



6.2.1 Check drawings

Before writing the program, make sure the job drawing contains all dimensions and technical requirements.

6.2.2 Tool configuration

According to the different workpiece, choose the appropriate tool, and then determine the processing sequence, combined with the machine tool to determine the installation position of the tool, tool shape. And draw the tool configuration diagram.

6.2.3 Processing configuration

According to the need, record the machining sequence, the number of each tool, cutting path in detail on the machining configuration chart.

According to the material material, outside diameter, type of machining tool and feed set cutting conditions.

6.2.4 Coordinate value

Read the key points of each tool path according to the workpiece drawing and machining configuration.

Since the X coordinate is the diameter coordinate, note that the X coordinate value is double.

6.2.5 programming

Write the program according to the machine instruction table and read coordinate values.

6.2.6 Processing data

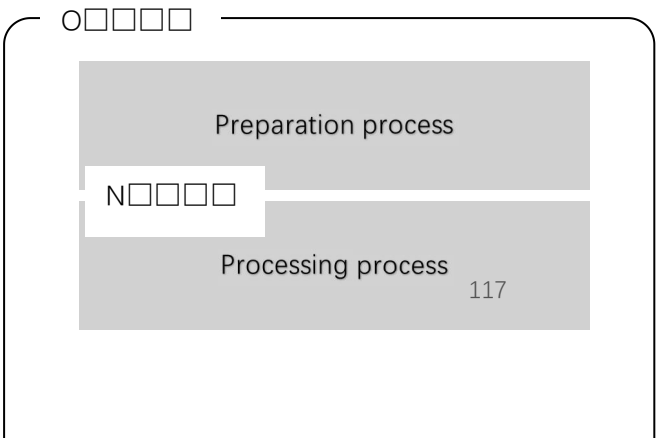
Machining data is the key data necessary for automatic operation and preparation of machine tools. Be sure to set the correct processing data for the program. Incorrect machining data may cause the machine to not run correctly or malfunction.

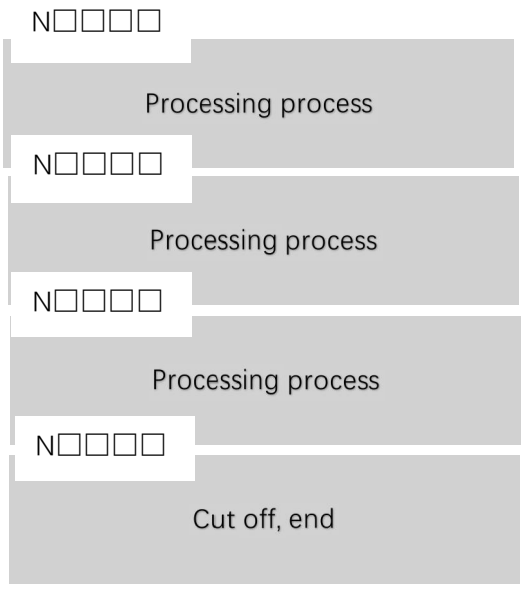
Again, adjust the tool data correctly.

6.3 Program structure

Good program structure can make the program easier to understand and debug, reduce the probability of error. Therefore, in the actual programming process, it is necessary to follow the fixed program structure.

The following is an example of a common program structure:





The procedure begins with the preparation process and ends with the cutting and closing process. Each processing procedure is between the preparation procedure and the finishing procedure. In addition, programs must start and end in the same place. From the beginning to the end of a program is called a single loop. The program consists of the following elements:

- **Program name:**O□□□□ in the figure above, you can define different program names based on the company's internal management regulations or picture numbers.
- **Sequence number:** N□□□□ in the figure above. The sequence number does not cause any operation of the machine tool. It is only used to indicate the program structure for easy understanding and logical control. For example, you can specify a combination such as N0101, where the first two digits represent the engineering sequence and the last two digits represent the tool number.
- **Segment:** An act of a program is a segment.
- **Single section:** Each program instruction and its subsequent value form a single section.
- **Segment skip:** A segment of a program that begins with a slash (/). This instruction is ignored when segment skip is enabled.
- **First zero:** If the number is a value other than "0", the previous "0" before the decimal point can be omitted, the leading "0" of the integer can also be omitted, and the "0" at the end of the value after the decimal point can also be omitted.
- **Instruction type:** See the table below for each instruction type:

Instruction code	Description	Instruction length, unit
G	Functional instruction	3-bit
N	Sequential instruction	6-bit
X、U	X1, X2 axis movement instruction	0.001mm/0.0001inch（Diameter value）

Y、V	Y1 axis move instruction	0.001mm/0.0001inch（Diameter value）
Z、W	Z1, Z2 move instruction	0.001mm/0.0001inch
C、H	Spindle indexing instruction	0.001°
R	Specified radius	0.001mm/0.0001inch
I、J、K	Specifies the center of the arc	0.001mm/0.0001inch
,A	Line Angle instruction	0.001°
,C	Even part chamfer	0.001mm/0.0001inch
,R	Even rounded corners	0.001mm/0.0001inch
F	Feed speed command	Every turn in feed 0.0001/rev、0.00001inch/rev Feed per minute 0.001mm/min、0.0001inch/min
S	Specify spindle rotation speed	5-bit rpm
T	Tool and tool compensation number instruction	4-bit
M	Auxiliary function instruction	3-bit
U、P、X	Designated pause time	0.001s~99999.999s（U、X）
P	Specifies the subroutine number	8-bit
L	Specifies the number of times the subroutine returns	4-bit
P、Q	Specified sequence number	6-bit

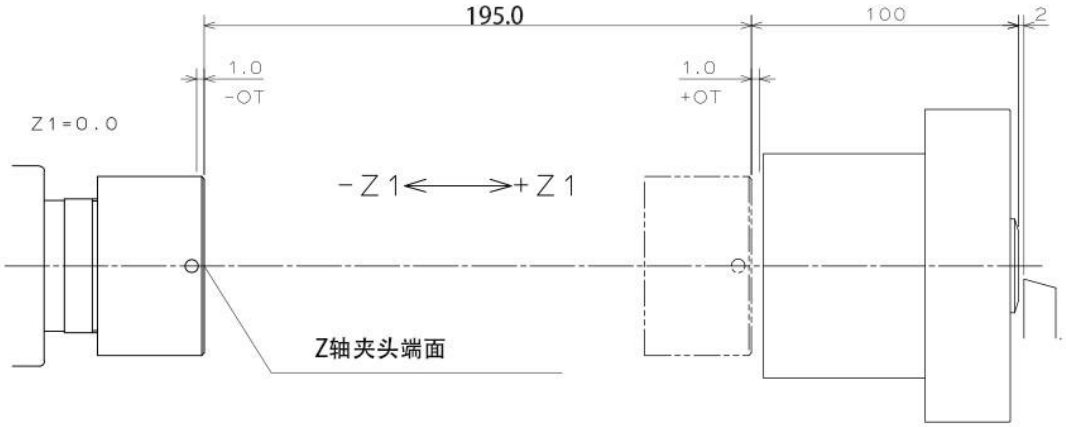
- **Decimal point:**The decimal point is only part of the instruction available, see the table below:

Unit	**Available instruction**
Distance mm、inch	X、Y、Z、U、V、W、I、J、K、R、,C、,R
Speed mm/rev 、mm/min inch/rev、inch/min	F
Time s	U、X
Angle °	C、H、,A

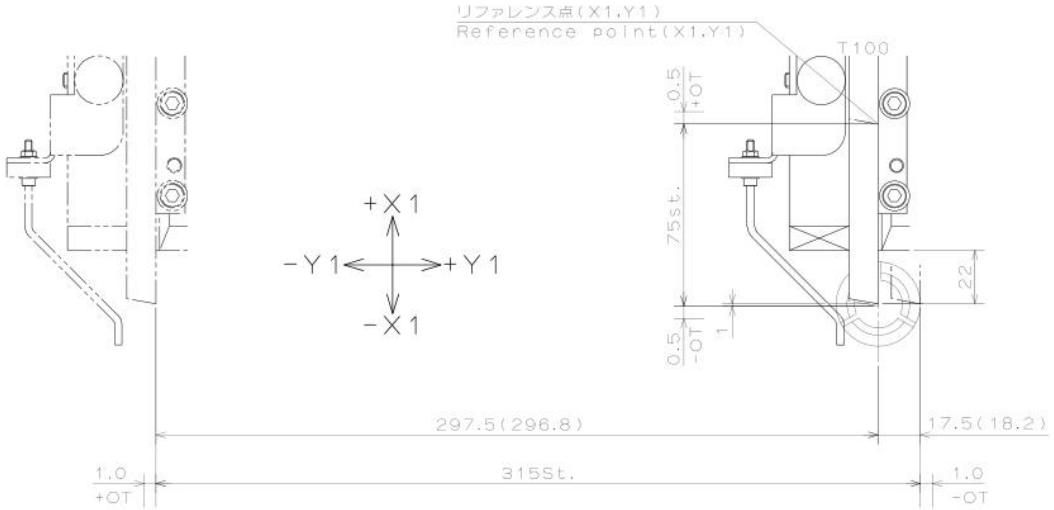
6.4 Coordinate system and travel of each axis

In the mechanical coordinate system of the machine, axes X1, X2 and Y1 take the center of the diameter of the material as the origin, axis Z1 takes the end face of the material at the end of cutting as the origin, and axis Z2 takes the reference point as the origin.

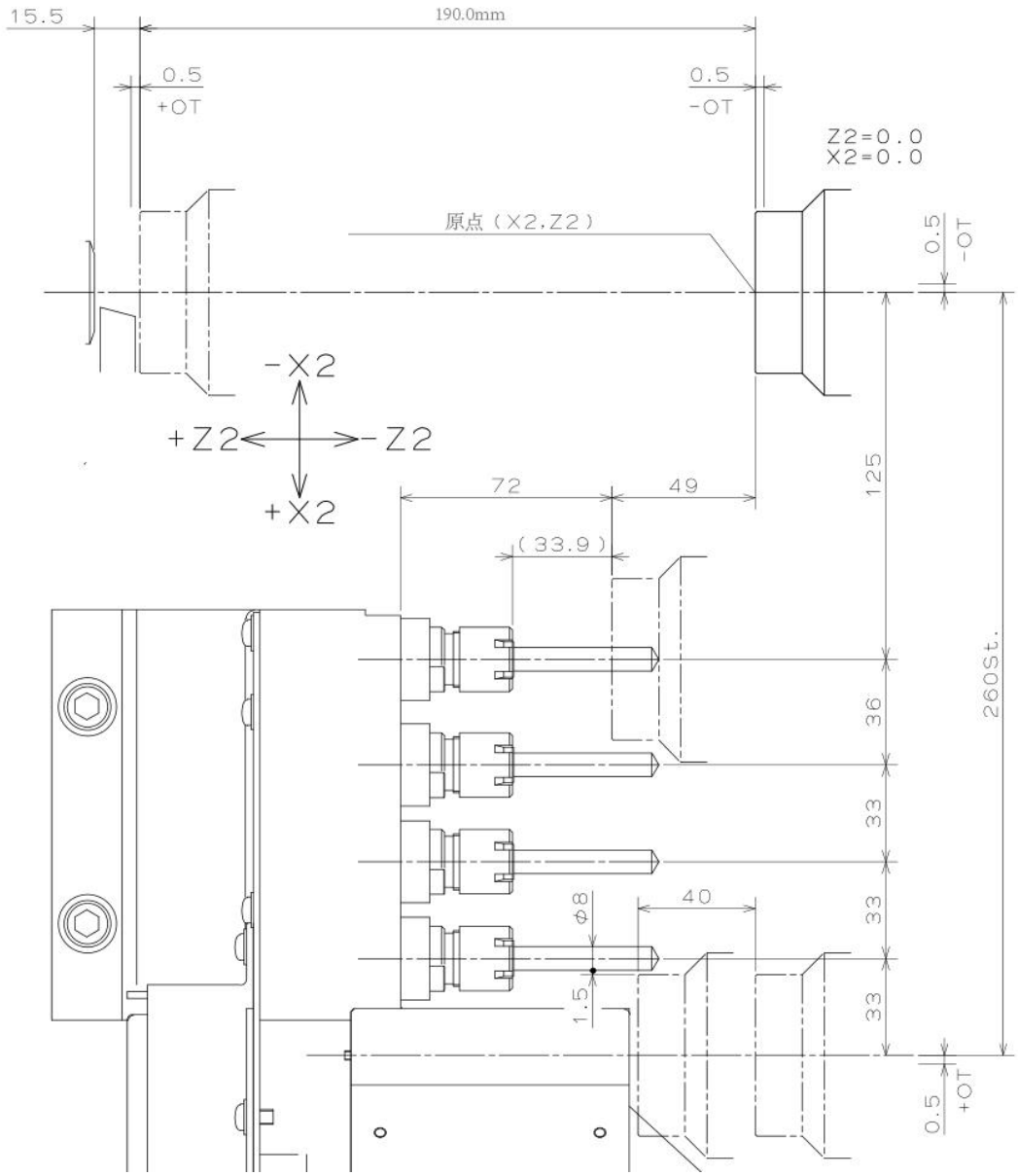
- Z1 axis


195.0
100
2
1.0
-OT
1.0
+OT
Z1=0.0
-Z1
+Z1
Z轴夹头端面

◆ X1、Y1 axis


リファレンス点(X1,Y1)
Reference point(X1,Y1)
T100
0.5
+OT
75St.
22
0.5
-OT
1
+X1
-Y1
+Y1
-X1
297.5(296.8)
17.5(18.2)
1.0
+OT
315St.
1.0
-OT

◆ X2、Z2 axis


15.5
190.0mm
0.5
+OT
0.5
-OT
Z2=0.0
X2=0.0
原点（X2,Z2）
0.5
-OT
-X2
+Z2
-Z2
+X2
72
49
125
(33.9)
260St.
36
33
33
33
40
φ8
1.5
0.5
+OT

6.5 Function instructions (G code)

G code list

G code	Function	Remarks
G00	Linear fast positioning	
G01	Straight cutting	
G02	Circular cutting (clockwise)	
G03	Circular cutting (counterclockwise)	
G04	Pause specified time	
G04.1	Intersystem wait	
G07.1	Cylindrical interpolation	
G09	Accurate location detection	
G10	Programming data entry	
G12.1	Priming pole interpolation	
G13.1	Cancel polar interpolation	
G17	Set the X-Y working plane	
G18	Set the Z-X working plane	
G19	Set the Y-Z working plane	
G20	Processing in British units	
G21	Processing in metric units	
G28	Reference point recovery	
G29	Revert from the reference point	
G30	Regression at any reference point	
G31	Jumping function	
G32	Thread cutting	
G34	Variable pitch thread cutting	
G40	Tool radius compensation cancels	
G41	Tool radius compensates to the left	
G42	The tool radius is compensated to the right	
G50	Coordinates set/maximum spindle speed limit	
G50.2	Cancel polygon cutting	
G51.2	Start polygon cutting	
G52	Local coordinate setting	
G53	Mechanical coordinate positioning	

G code	Function	Remarks
G54~G59	Working coordinate system setup	
G65	User macro simple call	
G66	User macro modal call	
G67	User macro modal call cancellation	
G70	Finishing cycle	
G71	Lateral (outside diameter) rough turning cycle	
G72	Radial (face) rough turning cycle	
G73	Shape contour rough turning cycle	
G74	End face (Z axis) peck machining cycle	
G75	Horizontal (X axis) pecking processing cycle	
G76	Compound thread cutting fixed cycle	
G80	Hole cycle cancellation	
G83	Front drilling cycle	
G84	End face (Z axis) tapping cycle	
G85	Front boring cycle	
G87	Side drilling cycle	
G88	Lateral (X axis) tapping cycle	
G89	Side boring circulation	
G90	Outside (inside) diameter turning cycle	
G92	Thread turning cycle	
G94	End turning cycle	
G96	Enable equal surface linear speed control	
G97	Cancel equisurface linear velocity control	
G98	Feed per minute (mm/min)	
G99	Feed per revolution (mm/rev)	
G113	Concurrent function cancellation	
G114.1	Spindle simultaneous function	
G114.3	Spindle bearing function	
G150	Z1 axis coordinate system setting	
G299	Processing data setting	
G300	Workpiece length setting	

6.6 Auxiliary instructions (M code)

M Code list

M code	description	System one($1)	System two ($2)
M00	Program pause	support	Do not support
M01	Selective program pause	support	Do not support
M02	Program stop	support	Do not support
M03	Positive spindle rotation	support	support
M04	Spindle reversal	support	support
M05	Spindle stop	support	support
M07	Reserve blow	support	Do not support
M08	Pump on	support	support
M09	Water pump shutdown	support	support
M10	Spindle tightening	support	support
M11	Spindle loosening	support	support
M16	Back shaft blow open	support	support
M17	Back shaft blow off	support	support
M26	The discharge machine is turning	support	support
M27	The discharge machine reverses	Do not support	Do not support
M28	The discharge machine stops	support	support
M30	The program ends and reverses	support	support
M33	Positive side power tool turn	support	support
M34	Side power tool reversal	support	support
M35	Side power tool stop	support	support
M43	Back power tool positive turn	Do not support	support
M44	Back power tool reversal	Do not support	support
M45	Back power tool stop	Do not support	support
M51	Refueling test	support	Do not support
M52	The output of the feeder stops opening	support	Do not support
M53	The output of the feeder stops and closes	support	Do not support
M54	The feeder is allowed to open backward	support	Do not support
M55	The feeder is allowed to shut down the day after tomorrow	support	Do not support

M code	Description	System one($1)	System two($2)
M56	Wait for refueling to finish starting	support	Do not support
M57	Spare function of the feeder	support	Do not support
M58	The back shaft is pushed forward	support	support
M59	Back shaft jacking back	support	support
M70	Specify the spindle as the machining spindle	support	support
M71	Specify the side power tool as the machining spindle	support	Do not support
M74	Specify the back shaft power head as the machining spindle	Do not support	support
M75	Z-axis load detection is on	support	support
M76	Z-axis load detection is off	support	support
M80	C1 axis indexing function is enabled	support	Do not support
M81	C1 axis indexing function is off	support	Do not support
M82	C2 axis indexing function is enabled	Do not support	support
M83	C2 axis indexing function is off	Do not support	support
M88	Multiplier 100% lock open	support	support
M89	Multiplier 100% lock off	support	support
M97	Tool life count	support	support
M98	Call subroutine	support	support
M99	End of subroutine	support	support
M103	The dorsal axis is positive	support	support
M104	Back axis inversion	support	support
M105	Dorsal stop	support	support
M110	Back shaft tightening	support	support
M111	Back axle release	support	support
M120	Confirm speed arrival	support	support
M121	Standby 1 is output ON	support	support
M122	Standby 1 outputs OFF	support	support
M123	Standby 2 is output ON	support	support
M124	The output of standby 2 is OFF	support	support
M125	Standby 3 output ON	support	support
M126	The output of standby 3 is OFF	support	support
M127	Standby 4 is output ON	support	support
M128	Standby 4 output OFF	support	support
M129	Standby 5 output ON	support	support

M130	Standby 5 output OFF	support	support
M150	Allow the spindle to start when the chuck is opened	support	support
M151	Do not start the spindle when the chuck is open	support	support
M158	Interference nullity	support	support
M159	Interference effectiveness	support	support
M340	Cut-off detection	support	Do not support
M1000	Start C1/C2 coupling	support	support
M1001	Turn off the C1/C2 coupling	support	support
M2000	Start the Z1/Z2 coupling	support	support
M2001	Disable the Z1/Z2 coupling	support	support

6.7 Tool instruction (T code)

T Code list

T code	Function	Remarks
T01	Disconnect detection switch No. 1	
T02	Row turning tool No. 2	
T03	Row tool turning tool No.3	
T04	Row tool turning tool No.4	
T05	Row tool turning tool No.5	
T06	Row tool turning tool No.6	
T07	Side rotary cutter No.7	
T08	Side rotary cutter No.8	
T09	Side rotary cutter No.9	
T10	Side rotary cutter No. 10	
T11	Front drilling tool No. 11	The main shaft and auxiliary shaft can be called
T12	Front drilling tool # 12	The main shaft and auxiliary shaft can be called
T13	Front drilling tool No. 13	The main shaft and auxiliary shaft can be called
T14	Front drilling tool No. 14	The main shaft and auxiliary shaft can be called
T20	Spindle back axis concentric 20	
T21	Back drilling tool # 21	
T22	Back drilling tool # 22	
T23	Back drilling tool # 23	
T24	Back drilling tool # 24	

T25	Back discharge	
T31	Face rotary tool No. 31	
T32	Face rotary cutter No. 32	
T33	Face rotary cutter No. 33	

Chapter Ⅶ Description of G code

7.1 Quick locating（G00）

Explanation: When the tool needs to move quickly to the specified coordinate system position (X, Z) for positioning. This command is used when returning the main axis.

Format:

G00 X(U)·····Y(V)·····Z(W)·····C(H)····· ;

X,Y,Z,C Absolute coordinate instruction	Moved position coordinates (end coordinates)
U,V,W,H Relative coordinate instruction	The distance from the current position as the reference direction of movement

For example:

G00 X10.0 ; X moved to position 10.0.

G00 U-5.0 Z20.0 ; When X moves to position 5.0, Z moves to position 20.0.

G00 X10.0 W30.0 ; When X moves to 10.0, Z moves to position 50.0.

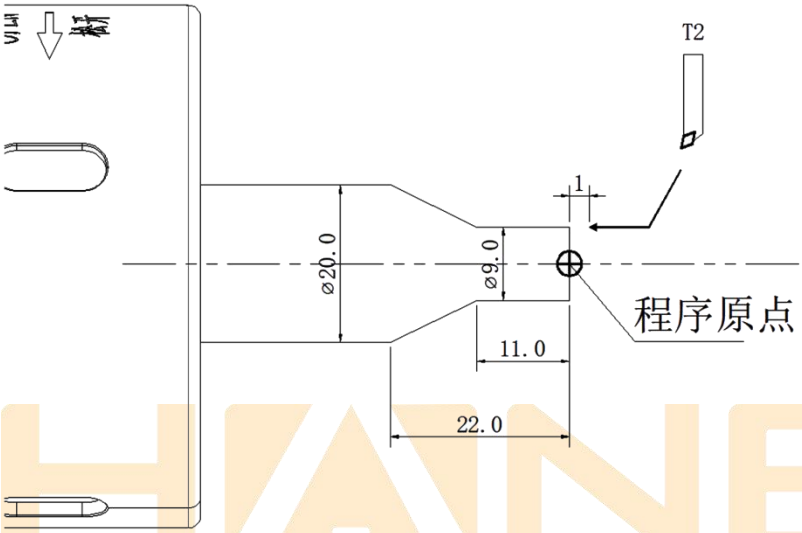
7.2 Straight line interpolation（G01）

Explanation: From the present position to the program position, when the tool moves in a straight line (cutting) at the feed rate of the F instruction.

Format:

G01 X(U)·····Y(V)·····Z(W)·····F ;

X , Y , Z , Absolute coordinate instruction	Moved position coordinates (endpoint coordinates)
U , V ,W, Relative coordinate instruction	The distance from the current position as the reference direction of movement
F	Feed rate



Example program:

```
T0202
M03 S1=2000 G99
M70
G0 X30.0 Z-8.0
G0 X9.0 Z-5.0
Z-1.0
G1 Z11.0 F0.05
G1 X20.0 Z-22.0 F0.035
```

7.3 Circular interpolation（G02 G03）

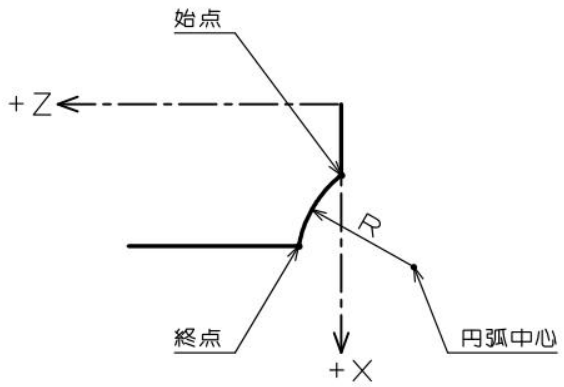
Explanation: From the present position to a certain position, the tool moves in a radius (cutting) at the feed speed of instruction F.

⚠PAY ATTENTION

- When the arc interpolation instruction is carried out, the plane selection instruction (G17,G18,G19) needs to be carried out in advance to determine the machining plane.
- G18(Z-X plane) when power input.
- R (the radius of the circular arc value) of instructions for less than 180 °. When the arc in more than 180 ° instructions, please separate two sequence of program instructions.

Format:

G2 / G3	X(U)_ Z(W) _ R _ F_ ;
G2 / G3	Y(V)_ Z(W) _ R _ F_ ;
G2 / G3	X(U)_ Y(V) _ R _ F_ ;

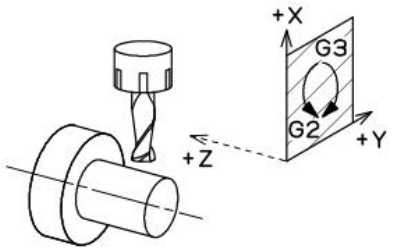
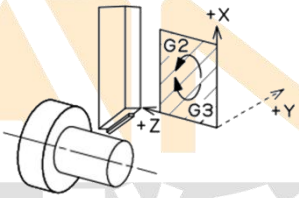
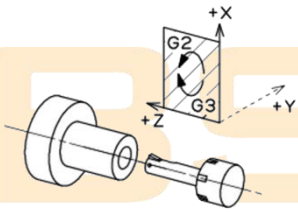
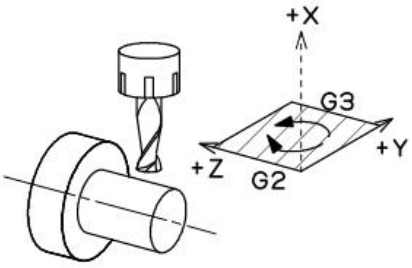
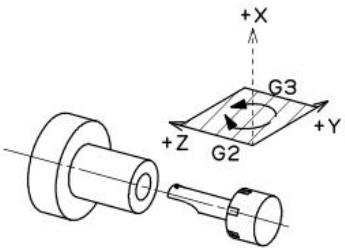


X,Z： The end coordinates of the arc

R： The radius of the arc

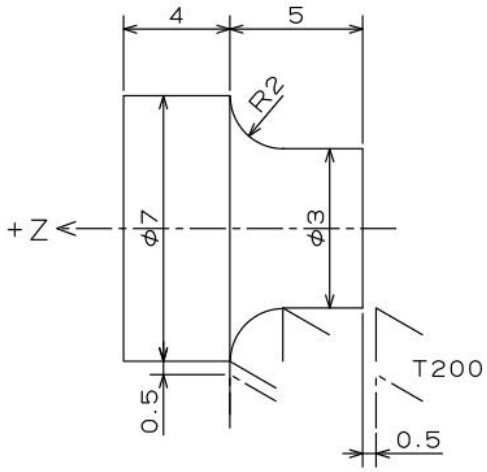
F： The feed velocity moving along the arc

Tool / Plane	T0100-T0600 T0700-T1000	T1100-T1400

X－Y plane （G17）	G17 ; G2 X(U) Y(V) R F ; G03	
		
Z－X plane （G18） Power input time	G18 ; G2 X(U) Z(W) R F ; G03	G18 ; G2 X(U) Z(W) R F ; G03
		
Y－Z plane （G19）	G19 ; G2 Y(V) Z(W) R F ; G03	G19 ; G2 Y(V) Z(W) R F ; G03
		

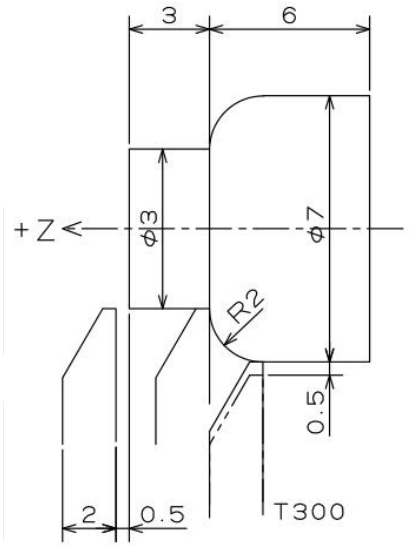
◇Working drawing

(1)



Program example
($1)
T0202;
M3 S1=3000 G99 ;
G0 X8.0 Z-0.5 ;
G0 X3.0 ;
G1 Z3.0 F0.03 ;
G3 X7.0 Z5.0 R2.0 F0.02 ;
G1 X8.0 F0.05 ;
G0 T0 ;

(2)



Program example
($1)
T0303;
M3 S1=3000 G99 ;
G0 X8.0 Z6.0 ;
G1 X7.0 F0.03 ;
G2 X3.0 Z8.0 R2.0 F0.02 ;
G1 Z11.5 F0.03 ;
G1 X8.0 F0.1 ;
G0 T0 ;

Z Dimensions have been added to the blade width(2㎜)。

⚠ PAY ATTENTION

- In the process of arc interpolation machining, plane selection instruction should be carried out in advance.

(G18 when power input: select Z-X plane.) G18(Z-X plane) when power input.

7.4 Pause（G04）

Explanation: After the time of the instruction in the time relay, the next program segment is executed.

Format:

G4 U .....; or G4 X..... ; or G4 P..... ;

U,X,P	Assignment of time After U,X,P instruction time to enter the next program segment. Because G4 is a modeless code instruction, it is only valid in this program segment. The specified time ranges from 0.001 to 99999.999 seconds. Address characters U and X, you can enter the decimal point in seconds. The address character P cannot enter a decimal point.

Example: Pause for 0.5 seconds

G4 U0.5 ; or G4 X0.5 ; or G4 P500 ;

7.5 Synchronization wait（G04.1）

Explanation: With this instruction, it can be linked to the ongoing running of the program in other systems.

Format: G04 .1 P1;

◇argument

G04.1 :Wait for instructions for synchronization

P :Identification number of confluence.(0 ~ 999)

Program example:

系统间同步等待

$1 (主轴程序)		$2 (背轴程序)	
G04.1 P1	等待	.	
.		G04.1 P1	等待
.		.	
.		.	
.		.	
.		.	
.		.	
.		G04.2 P2	等待
G04.2 P2	等待	.	
G04. 1P3	等待	.	
.		.	
.		.	
.		G04.1 P3	等待
G04.1 P4	等待	G04.1 P4	等待

⚠PAY ATTENTION
➢ If an error occurs in the confluent number, the system will issue an alarm (error in the order of the G04.1P argument).

7.6 Cylindrical interpolation（G07.1）

Note: G07.1 is used to start the cylindrical difference.

Because the center vector of a circle is not easy to calculate, the feature of this function is to convert the movement of the rotation axis specified by the Angle into the movement of the circumference, and carry out linear interpolation and circular interpolation between other axes. The G07.1 feature allows side programming of unrolled cylinders, making it easy to create cylindrical CAM slotting programs.

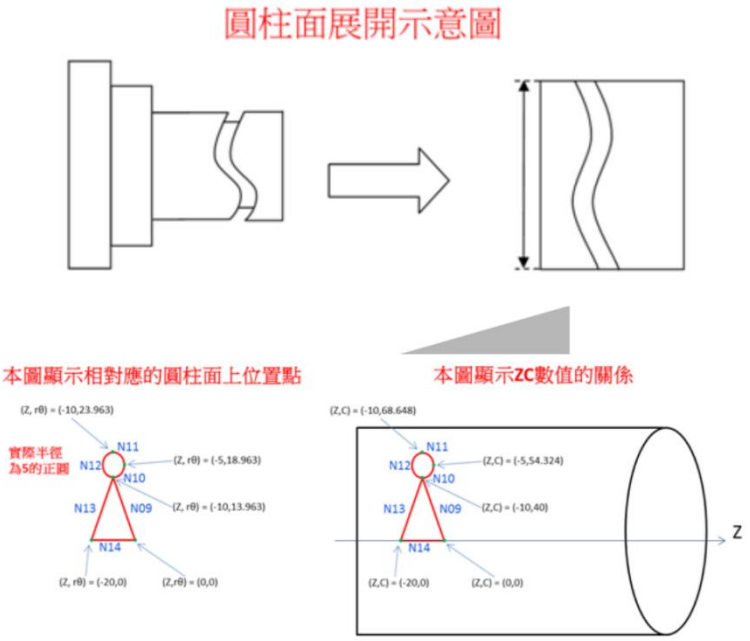
⚠PAY ATTENTION

➢ when G07.1, can use the cutter radius compensation (G40 / G41, G42).
➢ when G07.1, rate of feed mode, please switch to G98 mode first.
➢ Preparation for G02/G03 at G07.1: G02/G03 Z_C_ R_" Not provided "to centre vectorvalue: G02/03 Z_C_ I_J_(K_).
●Example: G3 Z-18.C180.R7.F100; // Use R radius.
➢ Declare the G19 plane before using G07.1
●Example: G19 C0 Z0;
➢ When using G07.1, (G00) is encountered with instructions related to fast positioning (e.g., positioning related: G28, G53, loop related :G70-76, G80-89... Etc.), because the system changes the coordinate system from the rotation axis to the linear axis, the fast positioning speed will be different from that without the G07.1 mode, which may cause the positioning speed to be too fast or too slow. It is recommended that the user cancel the G07.1 mode before using the relevant instruction
➢ Preparation for G07.1 mode: Users are required to avoid using G54-G59.9 "Working coordinate system setting" and G50 "coordinate system setting" because the system changes the coordinate system from rotation axis to linear axis, so as to avoid coordinate confusion。

Format:

```
G19 Z0 C0;      // Select the CZ working plane
G07.1 C__;      // Start cylinder difference, C_ cylinder radius
、
、              // (Path description)
、
```

G07.1 C0; // End cylindrical difference



Program example

Cylindrical interpolation（G07.1）	
T0808	T8 milling cutter
M33 S3=3000	Power head turning positive
M80	C1 axis start
M71	Specify the third spindle as the machining spindle
G00 X50.0 Z0	Fast positioning
G19 C0 Z0	Fast positioning of axis C
G07.1 C20.0	Start cylinder difference mode with cylinder radius value of 20.0
G42	Tip radius compensation
G98 G01 X40.0 F100.	Cut to bottom of cylinder
G01 Z-10.0 C40.0 F150.0	Triangle, first side
G03 Z-5.0 C54.324 R5.0	A quarter of a circle
G03 Z-10.0 C68.648 R5.0	A quarter of a circle
G03 Z-10.0 C40.0 R5.0	semicircle
G01 Z-20.0 C0.0	Triangle, second side
G01 Z0.0 C0.0	Triangle, third side
G40	Cancel tool radius compensation
G07.1 C0	Cancel cylindrical difference mode
G01 X50.0	Lift knife to set position

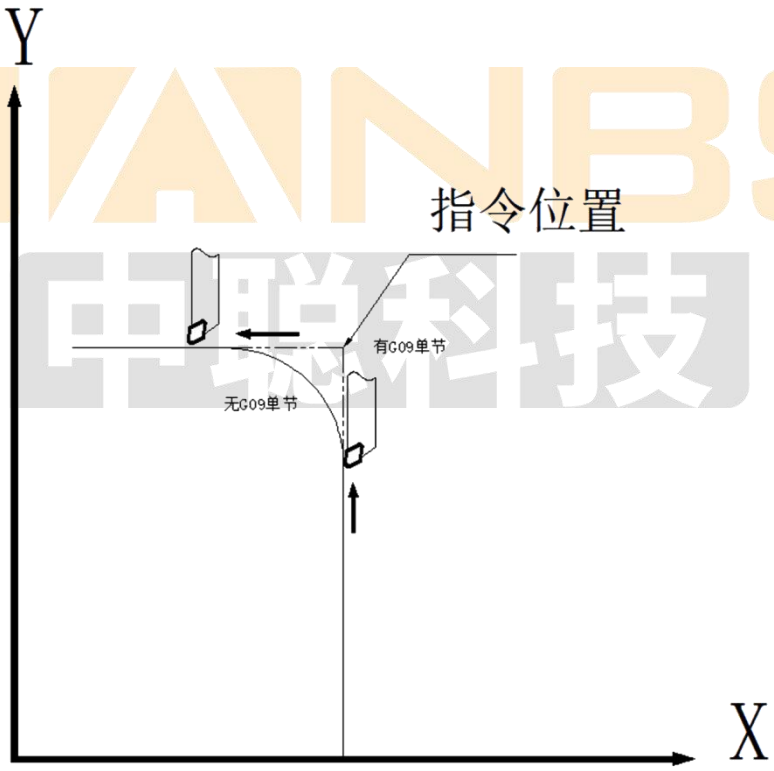
7.7 Do stop testing（G09）

Explanation: When processing the corner, sometimes because of the tool moving too fast or servo system delay relationship, can not really according to the shape of the corner cutting and cut into a little rounded corner, resulting in error, but in the case of requiring absolute right Angle accuracy, can use the G09 function to achieve, so that the tool near the corner deceleration, the position reaches a certain state (parameters set within the wide range) after confirmation, Each time a single section of instruction is executed.

Format:

G09 X__ Z__;

X、Z　　：Specify the angular position coordinates



7.8 Programming data input（G10）

Explanation: It can set and change the amount of tool wear correction.

When G10 instruction is carried out in absolute coordinates, the correction amount is

the new tool wear correction amount.

When the command is carried out in relative coordinates, the value after the adjustment of the acceleration and deceleration instruction is set as the adjustment of tool wear.

Format:

G10 P···X(U)···Y(V)···Z(W)···R(C)···Q···;

P	Tool wear correction number
X,Y,Z	Enter the absolute coordinate instruction value in the tool wear correction
U,V,W	Enter the increment of the relative coordinates in the tool wear complement
R	Input instruction knife tip R quantity. (Enter absolute coordinates)
C	The specified value is added or subtracted from the original tip R. (Enter relative coordinates)
Q	When the tip R is corrected, enter the position of the imaginary tip

For example: G10 P2 X0.1 W-0.035 ;

Subtract 0.035㎜ in Z complement

The X complement is set to 0.1㎜

Set tool wear correction number as "02"

7.9 Polar coordinate interpolation on/off (G12.1 G13.1)

1. Polar interpolation is the function of converting contour control from the command of a program in rectangular coordinates to a linear motion (
The motion of the tool) and the motion of a rotating shaft (the motion of the workpiece).
2. Polar coordinates interpolation plane, G12.1 start the polar coordinates interpolation mode and select a polar coordinates interpolation plane (as shown below).
Polar coordinate interpolation is completed in this plane.

Protection limitation:

1. Programming function of linear axis (X) diameter axis, effective version from 10.114.51, the selection is determined by PR4020:0 is radius axis programming; 1 Program the diameter axis.

2. Polar coordinate interpolation is cancelled after shutdown or system reset.

3. Coordinate display in polar coordinate interpolation mode: linear axis (X) and rotation axis (C) show the actual position with the radius axis.

4. The other axes show the actual position with the parameter setting.

5. The following G codes may be used in polar coordinate interpolation:

G01 linear interpolation

G02/G03 arc interpolation (IJR argument is the same as the general writing)

G04 Pause

G40/G41/G42 tool tip radius compensation

G65/G66/G67 user program call

6. Plane function (G17/G18/G19) is cancelled after polar coordinate interpolation is started; When polar coordinate interpolation is cancelled or the system resets, the system returns to the declared plane before startup.

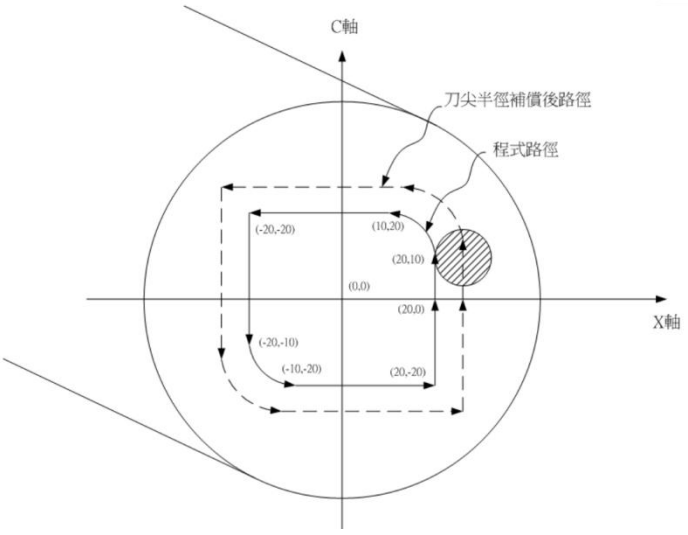
7. The coordinate system cannot be changed after polar interpolation is started (G50/G52/G53/G54~G59).

8. Polar coordinate interpolation cannot be started or canceled when the tool diameter compensation function is enabled (G41/G42), and can only be used when the tool diameter compensation function is canceled (G40).

9. After polar coordinates interpolation is started, if the tool diameter compensation function (G41/G42) needs to be enabled, it is necessary to add a single section with a knife movement of 0 to ensure the correct path.

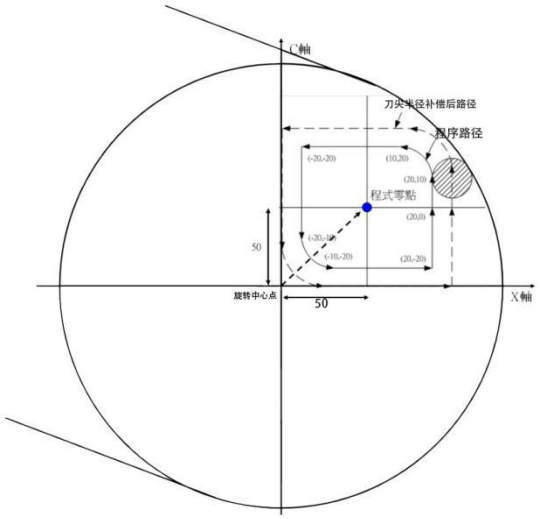
10. In polar coordinate interpolation mode, the tool diameter compensation cannot select the preview mode (PR3815=1). Program restart: For programs in G12.1 mode, the program cannot be restarted to avoid path errors.

11. After switching to polar coordinates, the action will be planned with the current Angle of C-axis as the imaginary 0 degree. Therefore, please perform the action of C-axis positioning before executing G12.1 to ensure the same feed Angle in the future. (Please refer to examples)



Program example

Polar coordinate interpolation（G12.1　G13.1）	
T3333	T33 milling cutter
M33 S3=3000	Power head turning positive
M80	C1 axis start
M71	Specify the third spindle as the machining spindle
G00 X55. C0. Z0	Fast positioning
G40 G98	Cancel tool compensation
G12.1	Priming pole interpolation
G41 G01 X50.	Tip radius compensation
G01 X20. F100.	
C10.	
G02 X10. C20. R10.	
G01 X-20.	
C-10.	
G02 X-10. C-20. R10.	
G01 X20.	
C0	
G13.1	Cancel polar interpolation
G40 G0 X55.	Lift knife to set position



Program example

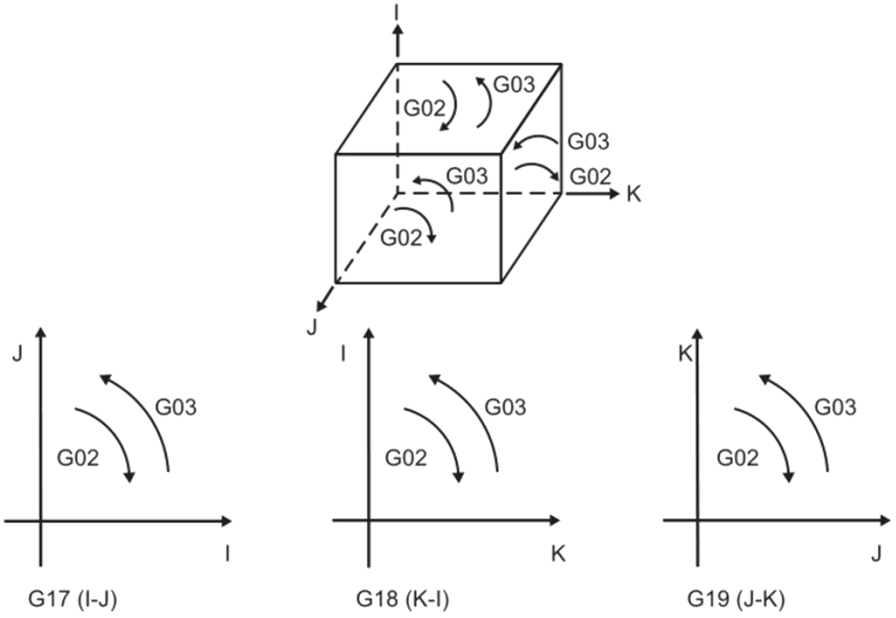
Polar coordinate interpolation（G12.1　G13.1）	
T3333	T33 milling cutter
M33 S3=3000	Power head turning positive
M80	C1 axis start
M71	Specify the third spindle as the machining spindle
G00 X70. C0. Z0	Fast positioning
G41 X55.	Cancel tool compensation
G01 X20. F100.	
C10.	
G02 X10. C20. R10.	Polar interpolation starts, eccentric (50, 50)
G01 X-20..	
C-10.	
G02 X-10. C-20. R10.	
G01 X20.	
C0	
G40 X55.	
G13.1	Cancel polar interpolation

7.10 Plane selection（G17,G18,G19）

Explanation: Plane selection (G17,G18,G19) is required when arc interpolation and tool tip R correction instructions are carried out.

Format:

code	Selective plane
G17	The X-Y plane
G18	Z-X plane (select when power is put in)
G19	The Y-Z plane



⚠PAY ATTENTION

- After the G18 instruction is executed as a separate program segment, the basic Z-X plane is returned.
- In a program segment without a plane-select instruction, the plane does not change.
- Since G18 is the return instruction for the basic Z-X plane of the circular interpolation plane, please be sure to process the engineering at G17,G19.
- Please cancel (G40) tool tip R correction (G41,G42) before carrying out plane selection instruction.

7.11 Setting and British and metric units（G20　G21）

G20：Imperial unit setting

G21：Metric unit setting

The coordinate origin offset of the workpiece, tool data, system parameters, and the position of the reference point are still correct after the system switch.

The system automatically handles unit conversion problems. After the British conversion, the following operating units will change with: display coordinates, rate units incremental inching units MPG inching units

◇ Format:

G20;

G21;

7.12 Automatic reference point retrieval （G28）

Explanation: After determining the position of the middle point of the instruction at the fast forward speed, it moves from the point at the fast forward speed (reset).

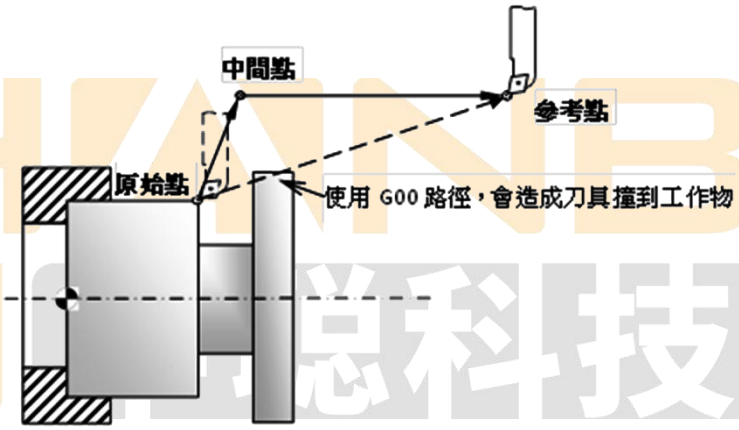
Format:

G28 U(X) V(Y) W(Z) ;

X、Z：Represents a specified intermediate point (in absolute value mode).

U、W：Represents a specified intermediate point (in incremental value mode).

diagram



⚠PAY ATTENTION

- When G28 instruction is executed, first move the tool to the specified intermediate point at the speed of G00, and then automatically return to the reference point (mechanical origin); Its main objective is to avoid the working object during the tool regression reference point so as not to cause the collision between the working object and the tool.
- Please give instructions after canceling the correction.

7.13 Reference point 2 restoration （G30)

After determining the position of the middle point of the instruction at the fast forward speed, it moves (reverts) from the point to the second reference point at the fast forward

speed.

(is a non-reference point and is the same as the reference point restoration using the G28 instruction, except that the second reference point position is returned).

◇format

G30 U(X)..... V(Y) .....W(Z) .....;

X、Z：Represents a specified intermediate point (in absolute value mode).

U、W：Represents a specified intermediate point (in incremental value mode).

⚠PAY ATTENTION

- G28 instruction execution, with the movement speed of G00, mobile tool to specify the intermediate point first, then automatically back to the reference point (mechanical origin); Its main objective is to avoid the working object during the tool regression reference point so as not to cause the collision between the working object and the tool.
- please cancel after receiving instructions.
- See 5-5 for usage.

7.14 Thread cutting （G32)

Explanation: Cutting of straight thread, cone thread, front thread and continuous thread;

Format:

G32 X(U) Z(W) F(E) Q ;

X(U)	Terminal coordinates
Z(W)	
F(E)	Pitch of thread (equivalent number of teeth per inch)
Q	The offset of the starting Angle of thread cutting. The command is used in the machining of multiple threads. The command is not required for single thread. Example 1: Q180000 is the thread cutting start Angle offset 180°. Example 2: Q180.0 is the thread cutting start Angle offset 180°.

(1)Parallel thread： G32 Z(W) Q (F___ or E___)；

(2)Taper thread： G32 X(U) Z(W) Q (F___ or E___)；

(3)Plane thread： G32 X(U) Q (F___ or E___)；

X、Z：Specify point position (absolute value mode)

U、W：Specify point location (incremental value mode)

F：Metric pitch

E：Inch pitch

Q：Thread begins to offset Angle, its function to make the cutting tool in every cutting to machining in rotation, the feed point are the same, used for wire thread turning more, generally turning single thread can be omitted, use the default value of Q = 0 ◦ (range: 0.001 ~ 360.000 ◦).

Figure (1) Single straight thread

Example program:
T0303 M03 S1=1000 G99 G4X0.3 G0X12.0Z-2.0 G0 X4.7 G32 Z6.5 F0.8 G0X8.0 Z-2.0 G0X4.5 G32 Z6.5 F0.8 G0X8.0 Z-2.0 G0X4.4 G32 Z6.5 F0.8 G0X8.0 Z-2.0 .

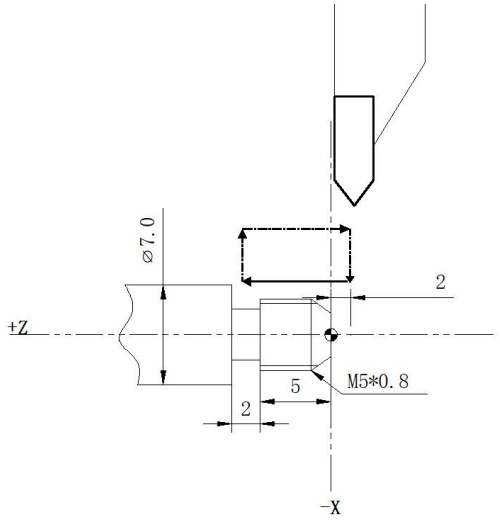


Figure (2) double thread taper thread

Example program:
T0303 M03 S1=1000 G99 G4X0.3 G0X12.0Z-2.0 G0 X3.7 G32 Z6.5 X4.7 F0.7 Q0 G0X8.0 Z-2.0 G0X3.7 G32 Z6.5 X4.7 F0.7 Q180.0 G0X8.0 Z-2.0 . . .

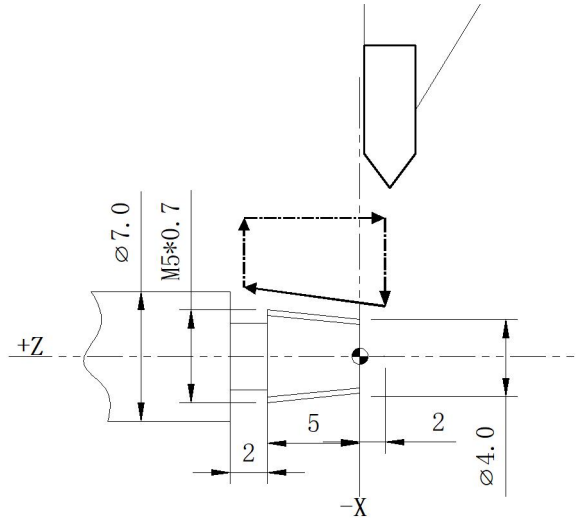
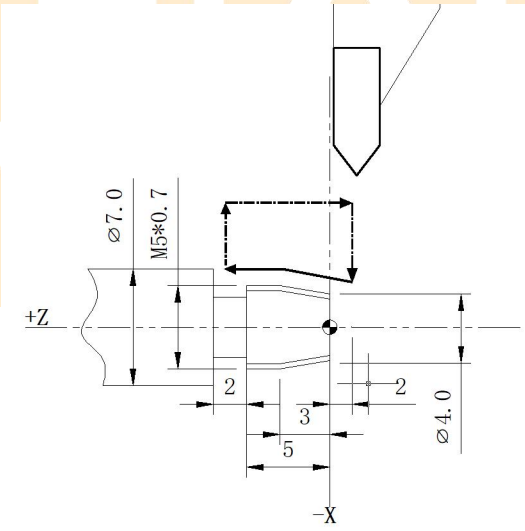


Figure (3) single thread continuous thread



Example program:

```
T0303
M03 S1=1000 G99
G4X0.3
G0X12.0Z-2.0
G0 X3.7
G32 Z3.0 X4.7 F0.7
G32 Z6.5 F0.7
G0X8.0
Z-2.0
G0X3.5
G32 Z3.0 X4.5 F0.7
G32 Z6.5 F0.7
G0X8.0
Z-2.0
.
.
.
```

⚠ PAY ATTENTION

- From rough cutting and fine cutting, spindle speed must be fixed.
- In thread cutting, if the use of pause, the thread will be damaged, so in thread cutting, can not use pause.
- Taper thread cutting instructions and spiral thread instructions can not be used at a certain speed.
- Do not thread cut G34 instruction outside G99 (in each Feed mode).

7.15 Cutting of variable pitch thread （G34）

Explanation: For variable pitch thread cutting instruction;

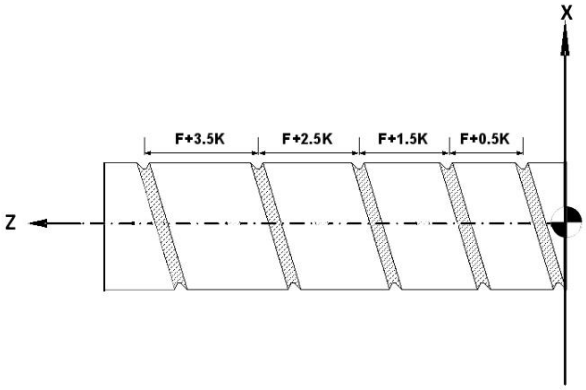
Format:

G34 X(U)..... Z(W) ..... F(E) ..... K .....;

X(U)	Terminal coordinates
Z(W)	
F(E)	Pitch at the starting point
K	The pitch of thread 1 turn is increased
Q	The offset Angle at the beginning of the thread

Example program:
F (pitch) : = 2.0
D (diameter) : =8.0
W (screw length) : =30.0

```
T0303
M03 S1=1000 G99
G4X0.3
G0X12.0Z-2.0
G0 X8.0
G34 W32.0 K1.0 F2.0
G0X12.0
Z-2.0
G0X7.8
G34 W32.0 K1.0 F2.0
G0X12.0
Z-2.0
```



Reference: 1. Same as G32 thread cutting, except for pitch increment (K) instruction.

2. When the input is reduced per transduction, so that the final pitch will be less than 0, the alarm of "car tooth distance can not be negative" will be generated; If the converted feed speed per minute passes the maximum cutting speed, the pitch will be reduced because the speed is limited, and a "tooth exceeds the maximum cutting speed" alarm will be generated.

3. Total movement in a single section: [F+(F+ winding number *K)]* winding number /2.

⚠PAY ATTENTION

- From rough cutting and fine cutting, spindle speed must be fixed.
- In thread cutting, if the use of pause, the thread will be damaged, so in thread cutting, can not use pause.
- Taper thread cutting instructions and spiral thread instructions can not be used at a certain speed.
- Do not thread cut G34 instruction outside G99 (in each Feed mode).
- Chamfering of threads is not allowed in G34 instruction.
- (For chamfer cutting, please use G92,G76 instructions).

7.16 Tool trip R correction（G40,G41,G42）

When taper and arc cutting are carried out with the tool with circular arc tip, the error can be calculated and corrected automatically if the tool position cannot be corrected.

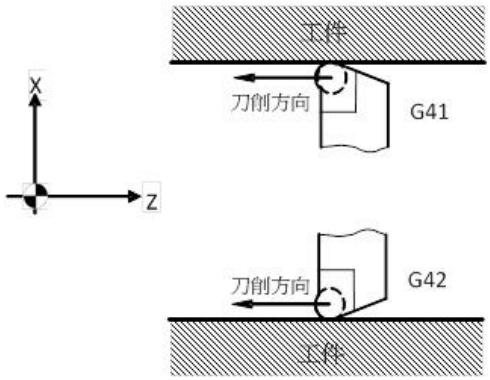
7.16.1 Z-X plane rectification method

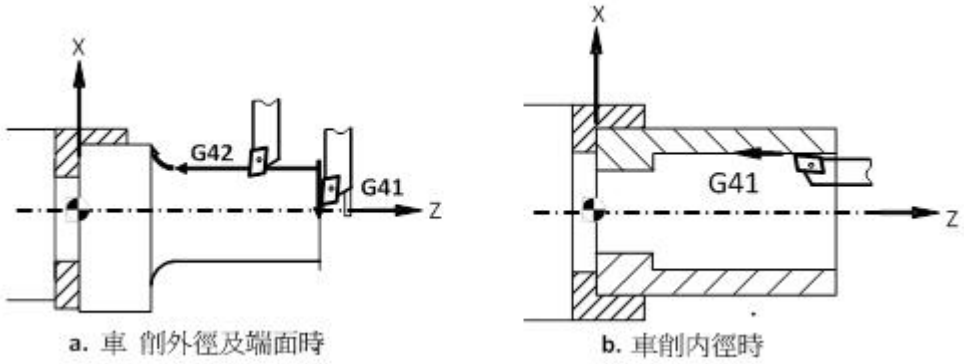
◇Instruction code

The processing position of the product surface from right to left is judged by the side diagram below the product center.

And corresponding to different directions of the cutting tool and different instructions on different sides of the raw material, to determine the direction of the correction of the tool with the arc tip.

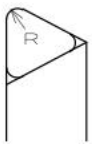
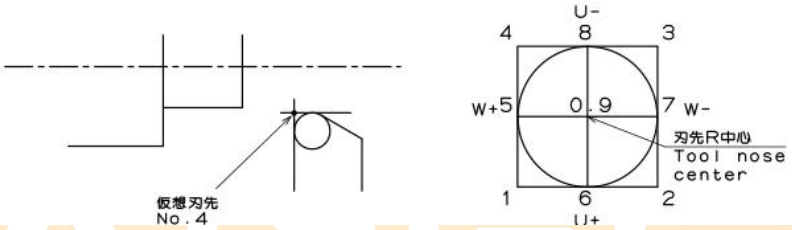
G code	Tool correction direction
G40	Cancel tip R correction
G41	The corresponding cutting tool direction is on the right side of the material
G42	The corresponding cutting tool direction is on the left side of the material





7.16.2 **Correcting login data**

Please input the size of the tool tip R and the number of the imaginary tool tip in the correction screen in advance;

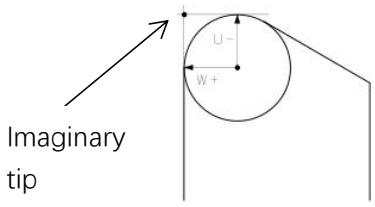
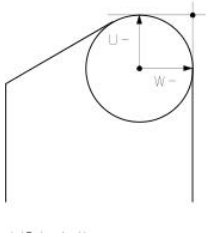
Tip R dimension R	Enter the size of the tool tip R in the address R of the tool wear correction number. 
Imaginary knife tip number T	Enter the imaginary tip number of the tool 0 to 9 in the address character T of the tool wear correction number of the tool. 

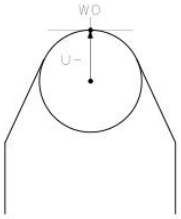
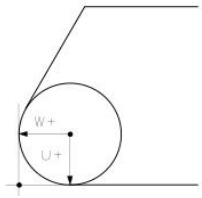
7.16.3 **Starting a program segment**

The program segment that starts with canceling the knife tip R and correcting G40 mode to G41 or G42 mode is called the start program segment. In this program segment.

Perform tip R correction and move the tool to the center right Angle of tip R in the next segment.

V, W is the direction of the imaginary knife point seen from the center of the knife point R, and the command is 1 or -1.

 G41 G1U-1.0 W1.0 F0.05	 G41 G1U-1.0 W-1.0 F0.05

 G41 G1U-1.0 W0.0 F0.05	 G42 G1U1.0 W1.0 F0.05

7.16.4 Canceling a program segment

G40 program segment is cancel program segment;

Please use the following format for instruction U, K.

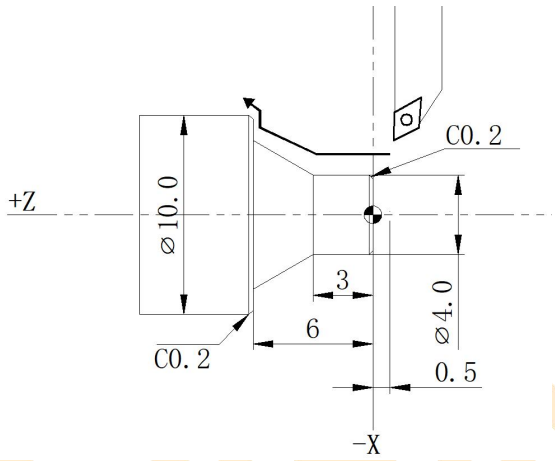
Axis	Outside diameter cutting		Bore cutting	
	G41	(G42)	(G41)	G42
X-axis	G40 U1 K1	G40 U1 K-1	G40 U-1 K -1	G40 U-1 K1

⚠PAY ATTENTION

- When the tool tip R is corrected and axis movement instruction for two consecutive program segments is not carried out, the action is the same as canceling.
- In G71 ~ G76 and G92 cycles, tool tip R correction was ineffective.
- When inputting instructions in MDI mode, the tool tip R correction is not performed.
- It is not allowed to process the inside of the circular arc with radius smaller than R of the tool tip.
- Do not process slots smaller than the diameter of the tool tip R.
- When machining the end face with radius smaller than R of the tool tip, please use a straight line for the instruction of this section difference.
- When the complement at the T code is 00, it is considered to cancel the segment.

7.16.5 Program examples

Outside diameter cutting



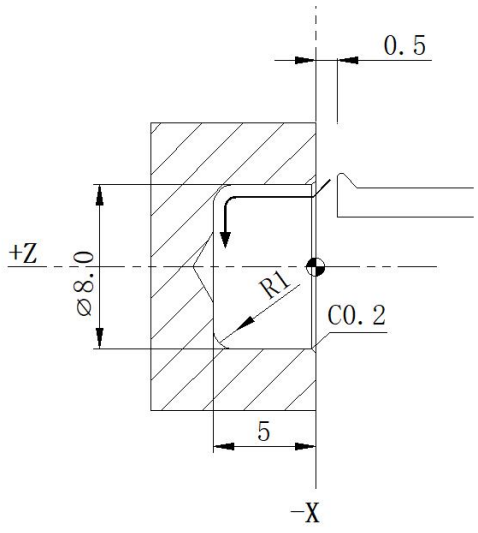
Tip R Dimension R = 0.2
Number the imaginary knife tip T = 4

〈Program example〉

```
T0202 M03 S1=2500 G99
G0 X11.0 Z-0.5
G1 X2.6 F0.2
G41 U-1 W1
G1 X4.0 Z0.2 F0.03
G1 Z3.0 F0.04
G1 X7.464 Z6.0
G4 U0.05
G1 X9.6 F0.05
G1 X10.6 W0.2 F0.03
G40 U1 W1 .0
G0 X11.0 T0
```

7.16.6 Program examples

Bore cutting



Tip R dimension R = 0.1
Suppose the knife tip number is T = 1

〈Program example〉

```
T1414 M3 S1=2000 G99
G0 X0.0 Z-5.0
G50 U3.0   (Tool width)
G0 X10.0
G42 G1 X9.4 Z-0.5
G1 X8.0 W0.7 F0.03
G1 Z4.0 F0.04
G2 X6.0 Z5.0 R1.0 F0.03
G1 X4.0 F0.04
G40 G0 Z-5.0
G50U-3.0
G0 T0 ;
```

7.17 Coordinate system setting and spindle locking speed setting（G50）

7.17.1 Setting the Coordinate system

◇Format:

G50 X(U)..... Y(V)..... Z(W)..... C(H)..... ;

X,Y,Z,C Absolute coordinate instruction	The absolute coordinate position is now the instruction value.
U,V,W,H Relative coordinate instruction	The absolute coordinate position is now only the offset value of the instruction value.

⚠PAY ATTENTION

- When executing the G50 instruction in the C-axis, please return the C-axis coordinate system to its original state in the position before the M9 instruction (C-axis OFF).
- G50 is a modeless G code that is valid only in this instruction program segment. Cannot execute other G code and M,S,T code instructions in the same program segment.
- Please give G50 instruction after canceling tool position correction and tool tip R correction.

7.17.2 Setting the spindle locking speed

When the cycle speed is controlled, the rotation speed of the shaft can be locked.

◇Format:

G50 S ····· ;

S	The upper limit of rotation speed specified The maximum rotation speed of the shaft is locked at the upper limit in the fixed cycle speed control (G96) instruction.

Example: G50 S2000; The maximum rotational speed of the locked shaft is 2000min-1.

7.18 Polygon cutting start/cancel（G50.2～G51.2）

⚠PAY ATTENTION

- Instruction G51.2 is to synchronously rotate the workpiece shaft and tool shaft, and execute polygon cutting at a certain proportional speed and phase difference
- Synchronous spindle speed: equal to the base spindle speed multiplied by Q divided by P.
- Synchronous phase difference: the Angle difference between the synchronous spindle and the base spindle rotating in the clockwise direction. If it is not less than R, phase synchronization will not be done
- Please refer to the new generation programming manual for specific use.

7.18.1 Turn on polygon cutting

G51.2 P_ Q_ [R_] [K_]

P	Basic spindle (workpiece shaft) speed ratio or blade number, using the internal value P=1(integer, range: 1~999).
Q	Synchronous spindle (tool shaft) speed ratio or edge number, use the internal value Q=1(integer, range: 1~999)
R	Synchronization phase difference (range: 0° to 359.999°).
K	Concurrent group number 1 to 3: Multiple concurrent groups can be used at the same time. A maximum of 3 groups can be used at the same time. (If K is not specified, the first concurrent group is used by default.)

7.18.2 Turn off polygon cutting

G50.2 [K_] ;

7.19 Bringing up the macro programs（G65～G67）

7.19.1 Simple program import

G65 P L Variable assignment;

G65 <文件名> L···· Variable assignment;

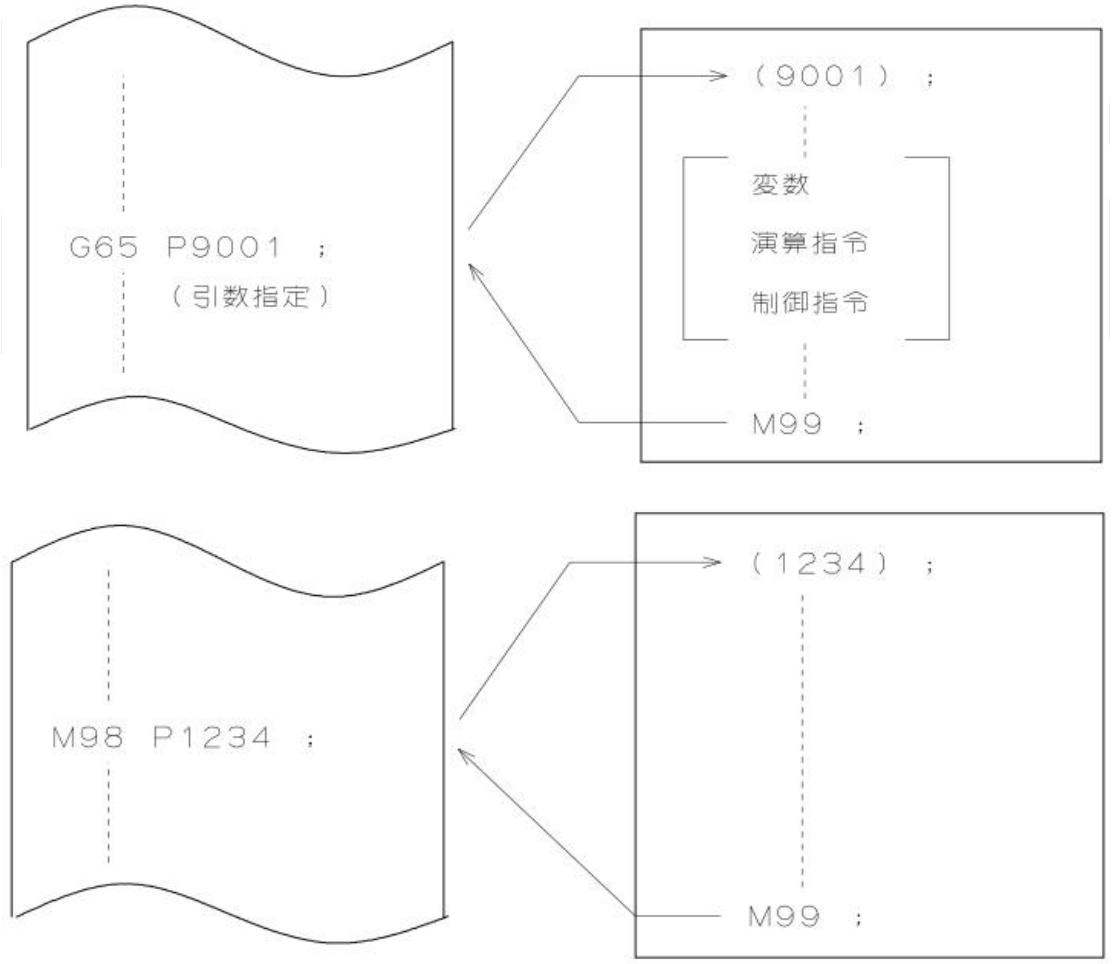
P	Call out procedure number;. Call up P specified program number macro program.

<File name>	Call up the file name; Please use <>.Enclose the file name;. The suffix of the file name contains a maximum of 32 characters.
L	Number of repetitions. Run this command for L times. L specifies that it is omitted once.
Variable	Variable designation. Specifies when a variable is specified within a macro program.

⚠PAY ATTENTION

- When a variable within a macro program needs to be specified, the variable is specified in < variable specified >.

Use 1 set of commands to make a special program and store it in program memory. This sequence of programs can be invoked and executed using G65 or G66 instructions. Calling the special program from the macro program ontology can be made into a macro program in the same way as the subroutine.



However, macroprograms differ from subroutines in the following ways.

You can use variables.

The "calculus" between variable and constant can be performed.

"Control instructions" such as conditional jumps can be used.

According to these functions, complex calculations and necessary judgments can be made in macro program ontology. So as to make a better universal program. When the macro program ontology is called from the main program, the actual value of the macro program ontology variable can be input according to the "variable specified", and the macro program executes a series of specific programs such as tool moving according to the result.

In this manual, there are cases where a single "macro" is called a macro program.

7.19.2 Importing macro commands

Call out the macro program ontology method, there are four kinds as follows;

№	Call-out method	Use code	Remark
1	Simple call out	G65	
2	Mode call	G66	G67: cancel
3	Call up any G code	G□□	
4	M code out	M□□	

7.19.3 Simple call out (G65)

G65 P‥‥L‥‥〈Variable assignment〉;

P: Call out program number.

L: The number of returns (1 if omitted).

·Execute this command to call up the specified program number in P macro program, and execute L times.

·When a variable within a macro program needs to be specified, the variable is specified in < variable specified >.

7.19.4 Mode call (G66, G67)

G66 P‥‥L‥‥〈Variable assignment〉;

P: Call out program number.

L: The number of returns (1 if omitted).

·Execute this command to call up the specified program number in P macro program, and execute L times.

・When a variable within a macro program needs to be specified, the variable is specified in < variable specified >.

G67 ;

・Execute this command to cancel the macro's mode call.

Reference: Macro program call (G65, G66) and subroutine call out (M98) difference is as follows:

- Macroprogram calls can specify variables (read data in macroprogram), but subprograms cannot.
- When there are other CNC instructions in the subroutine call out program segment, call out the subroutine after executing the instruction. But in macro program call out is not executed at all, only execute macro program call out.
- When there are other CNC instructions in the subroutine call out program segment, the single step segment can be executed to stop. Macros, on the other hand, cannot be stopped.

7.19.5 Specifying variables

Variables used in the macro ontology are called variables when they are actual values.

Therefore, variable assignment specifies that the variable is a real number.

There are two types of variables: | and ||, both of which can be used freely.

7.19.6 Variable specified |

Variables can be specified using all 21 address characters except G,L,N,O, and P.

If I,J, and K are used, they must be specified in alphabetical order. Other special address characters are not specified in alphabetical order.

7.19.7 Variable specified ||

A,B,C variables and I,J,K can be used to specify 10 groups of variables.

In this case, you need to specify the order I,J,K.

⚠PAY ATTENTION

- Variable assignment Ⅰ and variable assignment Ⅱ can be used together.
- When the same variable is specified repeatedly, the latter method is valid.
- In variable assignment Ⅰ, Ⅱ, please specify the address characters I,J,K in alphabetical order. Other address characters can specify the order freely.
- Variable designations are independent of address characters and can use symbols and decimal points.
- Be sure to do the G65,G66 instructions before all variables are specified.

7.19.8 Macro usage instructions

- The basic G, T and M codes built into the controller may not meet the needs of various industrial applications. In this regard, the new generation technology provides the function of custom macroset.
- Enable developers to develop the corresponding macro program according to the characteristics and requirements of the machine, provide the special action for the machine, improve the value of the machine.
- Before we start, let's start with a simple basic definition of how the main program calls other programs
- Call subroutine: The program that executes the call and cannot read the occupied arguments
- Call macros: Programs that execute calls and can read occupied arguments
- See Argument Description for the definition of arguments

7.19.9 Macros classification

- All macros need to meet the following conditions
  - Write in macros syntax
  - File starts with %@MACRO
  - No secondary file name at the end of the file name
  - File ending must be M99, back to the main program of the call
  - Macros can be basically divided into the following categories.
  - G-code macros
  - Modeless call Macros G code (G65)
  - Modal Call macros G code (G66/G66.1)
  - T-code macros
  - M code macros

7.19.10 Macros classification

- The G-code macros are used as examples below. If other macros have different specifications, they will be specifically explained.
- When the processor executes the G-code, it executes the contents of the G-code giant.
- The last line in the G-code macro writes M99. When the G-code macro is executed to M99, it will return to the main program to continue processing.
- Actions in the G-code macros may change the system state. For example, pressing G90/G91 in the G-code macros causes the command mode to change
- To avoid affecting the execution of the main program, or the operation of other macros/subprograms, the state of the main program is usually expected to be the same before and after the execution of the G-code macros.
- Therefore, it is recommended that the system state be backed up at the beginning of the G-code macros and restored at the end of the G-code macros.

7.19.11 public variables

- Universal variable （# 1 ~ #400）.
- System variable （@401 ~ @15535）Do not use.

⚠PAY ATTENTION

- All common variables have no lifetime limit unless they are parallel.
- If you still need to remember @1~@400 after shutdown, please use No3811 to set it.
- DOS supports only R0 to R7999, so be careful when using corresponding common variables.

7.19.12 Area variables

- Region argument variable (#1 ~ #26) A region variable that the system uses internally as a macroset argument.
- Common variable (#27 ~ #400) A region variable used by macros and subroutines.
- Do not use system variables (#1000 ~).

⚠PAY ATTENTION

- MACRO variables used in the area, its life cycle is limited to the MACRO executes, when finish 扒 line and leave the area into a VACANT state variables automatically reply.

- Deputy program variables can be Shared with his father program area, its life cycle in the parent program 执 followed at the line end.
- MACRO arguments table, see [Macro Considerations].

7.19.13 Operator

operator	symbol	Execution sequence
parenthesis	( ) [ ]	1
Function assignment	Identifier (argument)	2
Negative sign	-	3
complement	NOT	3
Multiplication sign	*	4
Division sign	/	4
modulus	MOD	4
Plus sign	+	5
Minus sign	-	5
compare	<,>,<=,>=	6
Equal to	=	7
Not equal to	<>	8
Boolean operation "and"	&,AND	9
Boolean operation "mutually exclusive"	XOR	10
Boolean operation or	OR	11

7.19.14 Syntax description

Grammar:GOTO

Explanation: Skip to the specified N line number. If there are two N line numbers in the program, the first N line number in the program prevails

Example program:

```
% @MACRO
#1 := 1;
#2 := 10;
G0 X0. Y0. ;
IF( #1 = 1 ) THEN
   GOTO #2;
END_IF;
IF( #1 = 2 ) THEN
   GOTO 100;
END_IF;
```

```
N10;
G0 X50. Y0. ;
N100;
G0 X0. Y50. ;
M30;
```

Reference: When using REPEAT/WHILE/FOR/GOTO and other loop functions, caution should be paid to the infinite loop problem. When this problem occurs, the man-machine picture will be locked and unable to operate or cause the processing program to be stuck. It is suggested to add the SLEEP() function in the loop at the right time, which can still operate the man-machine screen to abort the program execution when infinite loop occurs.

Grammar:CASE

CASE OF
<Integer>:
<Statement list>
<Integer>, <Integer>:
<Statement list>
<Integer>, <Integer>,Integer>:
<Statement list>
ELSE
<Statement list>
END_CASE;

Explanation:CASE Multi - condition judgment, according to the condition variable content, respectively execute different program blocks. Note that the contents of the variable must be integers

Example program:

```
% @MACRO
#1 := 1;
G0 X0. Y0.;
CASE #1 OF
1:
  G0 X(1.0*#1) Y(1.0*#1);
2:
  G0 X(2.0*#1) Y(2.0*#1);
3, 4, 5:
  G0 X(3.0*#1) Y(3.0*#1);
ELSE
  G0 X(4.0*#1) Y(4.0*#1);
END_CASE;
M30;
```

Grammar: IF

<condition> IF THEN
<Statement list>
<condition> ELSEIF THEN
<Statement list>
ELSE
<Statement list>
END_IF

Explanation: IF Conditional judgment

Example program:

```
% @MACRO
```

```
#1 := 3.0;
G0 X0. Y0. ;
IF #1 = 1 THEN
   G0 X(1.0*#1) Y(1.0*#1);
ELSEIF #1 = 2 THEN
   G0 X(2.0*#1) Y(2.0*#1);
ELSEIF #1 = 3 THEN
   G0 X(3.0*#1) Y(3.0*#1);
ELSE
   G0 X(4.0*#1) Y(4.0*#1);
END_IF;
M30;
```

Grammar: REPEAT

REPEAT
<Statement list>
<condition> UNTIL
END_REPEAT

Explanation: REPEAT Loop control

Example program:

```
% @MACRO
#10 := 30.;
#11 := 22.5.;
#12 := #10/2;
#13 := #11/2;
#14 := 2.0;
#15 := 1.5;
G00 X#12 Y#13 ;
REPEAT
   G00 X(#12+#14)   Y(#13+#15);
   G00 X(#12+#14)   Y(#13-#15);
   G00 X(#12-#14)   Y(#13-#15);
   G00 X(#12-#14)   Y(#13+#15);
   G00 X(#12+#14)   Y(#13+#15);
   #14 := #14 + 2.0;
   #15 := #15 + 1.5;
UNTIL (#14 > #12) OR (#15 > #13) END_REPEAT;
M30;
```

Grammar: WHILE

WHILE <condition> DO
<Statement list>
END_WHILE;

Explanation: WHILE Loop control

Reference: When using REPEAT/WHILE/FOR/GOTO and other loop functions, caution should be paid to the infinite loop problem. When this problem occurs, the man-machine picture will be locked and unable to operate or cause the processing program to be stuck. Recommend timely in circle back to join the SLEEP () function, but in the case of infinite circle back, still can operate man-machine to suspend the program 执 line picture.

Example program:

```
% @MACRO
#10 := 30.;
#11 := 22.5.;
#12 := #10/2;
#13 := #11/2;
#14 := 2.0;
#15 := 1.5;
G00 X#12 Y#13 ;
WHILE (#14 <= #12) AND (#15 <= #13) DO
   G00 X(#12+#14)   Y(#13+#15);
   G00 X(#12+#14)   Y(#13-#15);
   G00 X(#12-#14)   Y(#13-#15);
   G00 X(#12-#14)   Y(#13+#15);
   G00 X(#12+#14)   Y(#13+#15);
   #14 := #14 + 2.0;
   #15 := #15 + 1.5;
END_WHILE;
M30;
```

Reference: When using REPEAT/WHILE/FOR/GOTO and other loop functions, caution should be paid to the infinite loop problem. When this problem occurs, the man-machine picture will be locked and unable to operate or cause the processing program to be stuck. Recommend timely in circle back to join the SLEEP () function, but in the case of infinite circle back, still can operate man-machine to suspend the program 执 line picture

Grammar: FOR

FOR <variable 1> := Statement 1> <Statement 2> < Statement 3> TO BY DO
<Statement list>
END_FOR;

Explanation: FOR Loop control

variable 1: A variable that controls the number of loops.

Statement 1: The initial number of loop counts, which can be numerical value or expression

Statement 2: The number of terminations of the loop count, which can be a numerical value or an expression

Statement 3: The number of accumulative times of each cycle counting can be a value or an expression

Statement list: back to the circle line 执 content at a time

Example program:

```
% @MACRO
#10 := 30.;
#11 := 22.5.;
#12 := #10/2;
#13 := #11/2;
#14 := 2.0;
#15 := 1.5;
G01 G90 X#12 Y#13 F1000;
FOR #6 := 0 TO 3 BY 1.0 DO
   G00 X(#12+#14) Y(#13+#15);
   G01 X(#12+#14) Y(#13-#15);
   G01 X(#12-#14) Y(#13-#15);
   G01 X(#12-#14) Y(#13+#15);
   G01 X(#12+#14) Y(#13+#15);
   #14 := #14 + 2.0;
   #15 := #15 + 1.5;
END_FOR;
M30;
```

Reference: When using REPEAT/WHILE/FOR/GOTO and other loop functions, caution should be paid to the infinite loop problem. When this problem occurs, the man-machine picture will be locked and unable to operate or cause the processing program to be stuck. Recommend timely in circle back to join the SLEEP () function, but in the case of infinite circle back, still can operate man-machine to suspend the program 扒 line picture

Grammar: EXIT

Explanation: Loop break, jump out of loop control

Example program:

```
% @MACRO
#10 := 30.;
#11 := 22.5.;
#12 := #10/2;
#13 := #11/2;
#14 := 2.0;
#15 := 1.5;
#16 := 1.0;
G00 X#12 Y#13 ;
FOR #6 := 0 TO 3 BY 1.0 DO
   IF((#14 = 4) & (#16 = 1)) THEN
      EXIT;
```

```
    END_IF;
        G00 X(#12+#14)   Y(#13+#15);
        G00 X(#12+#14)   Y(#13-#15);
        G00 X(#12-#14)   Y(#13-#15);
        G00 X(#12-#14)   Y(#13+#15);
        G00 X(#12+#14)   Y(#13+#15);
        #14 := #14 + 2.0;
        #15 := #15 + 1.5;
END_FOR;
M30;
```

7.19.15 Monitoring area variable address

⚠PAY ATTENTION

- Please use (@1~@400) (#1~#400) for variables.
- (@401 to @165535) (#1000~) is for system use.
- Read the instruction manual carefully before using macro variables.

Explanation: Operators can program according to the variable address monitoring program execution content status

图 3



7.19.16 Argument description

- The argument uses 26 letters in English, and each letter except G/N/O has a corresponding program variable (# value), as shown in the table below

Argument	#	Argument	#	Argument	#
C	#3	L	#12	U	#21
D	#7	M	#13	V	#22
E	#8	N		W	#23
F	#9	O		X	#24
G		P	#16	Y	#25
H	#11	Q	#17	Z	#26
I	#4	R	#18		

7.19.17 Function table

Function	Description

ABS	Calculate the absolute value of a number
	Ex:
	#10 = -1.1;
	#1 := ABS(#10); // #1 = 1.1
	#2 := ABS(-1.2); // #2 = 1.2

ACOS	Calculate the inverse cosine of a number
	Ex:
	#10 := 1;
	#1 := ACOS(#10); // #1 = 0
	#2 := ACOS(-1); // #2 = 180

ALARM	Trigger a macro alert
	Ex:
	ALARM(300); // Trigger Macro Alert 300
	ALARM(301, "ALARM 301 Content");
	Alert content has string length limits
	English: 19 characters
	English: 39 words

ASIN	Calculate the arcsine of a number
	Ex:
	#10 := 1;
	#1 := ASIN(#10); // #1 = 90
	#2 := ASIN(-1); // #2 = -90

ATAN	Calculate the inverse tangent of a value Ex: #10 := 1; #1 := ATAN(#10); // #1 = 45

Function	Description
	#2 := ATAN(-1); // #2 = -45

CEIL	Returns the smallest integer greater than or equal to a value
	Ex:
	#10 := 1.4;
	#1 := CEIL(#10); // #1 = 2
	#2 := CEIL(1.5); // #2 = 2

CLOSE	Close the file previously opened by the OPEN function. The file will also close automatically when the program ends. When the file is closed, the PRINT function is invalid. For example: CLOSE(); // Close the file

COS	Calculate the cosine of a number
	Ex:
	#10 := 180;
	#1 := COS(#10); // #1 = -1
	#2 := COS(-180); // #2 = -1

DBLOAD	Read the data of the specified index from the currently loaded XML table. Please refer to the attachment for related applications.
	For example:
	DBOPEN("FLAT
	TAB01");
	// Load FLAT\\TAB01 file
	DBLOAD(0);
	// Read the 0th data
	DBLOAD(1);
	// Read the first data

DBOPEN	Loads the specified XML data table. The XML data table path is located at the GNCFILES location specified by the user. Please refer to the attachment for related applications.
	Example: DBOPEN("Test");
	// Load the GNCFILES\\Test file
	For example:
	#1 = 51;
	DBOPEN("FLAT\\\\AB#1[3]ZZ");
	// Load the file FLAT\\\\AB051ZZ where [3] indicates the file name is three significant digits

EXP	Calculate index values based on natural numbers. EX: #1:=EXP(1); // e^1 = 2.71828

FLOOR	Returns the largest integer less than or equal to a value
	EX:
	#10 := 1.4;
	#1 := FLOOR(#10); // #1 = 1
	#2 := FLOOR(1.5); // #2 = 1

Function	Description

GETTRAPARG	The arguments of G66/G66.1 in the Trap section are read
	For example:
	Assume that the contents of the O0001 main program are
	G66 P100 X100. Y100.
	G01 X20.
	The O0100 subroutine uses GETTRAPARG to read arguments
	#1 := GETARG(X);
	// Save the X argument 100. To #1
	#2 := GETTRAPARG(X);
	// Save the Trap section X argument 20. To #2

LN	Computes pairs based on natural numbers.
	EX:
	#2:=LN(100); // ln100 = 4.60517
	Note: The rear argument must be positive, otherwise the alarm will jump COR-121 LN() operand must be positive

MAX	Determines the maximum value of two input values
	Ex:
	#10 := 1.2;
	#20 := 4.5;
	#1 := MAX(#10, #20); // #1 = 4.5
	#2 := MAX(-1.2, -4.5); // #2 = -1.2

MIN	Determines the minimum value of the two input values
	Ex:
	#10 := 1.2;
	#20 := 4.5;
	#1 := MIN(#10, #20); // #1 = 1.2
	#2 := MIN(-1.2, -4.5); // #2 = -4.5

MSG	Customizing tips, please refer to MACRO Customizing Tips
	Ex:
	MSG(100); // Alarm ID
	MSG(" bit displacement "); // Alert display content
	MSG(100, "bit missing "); // Prompt ID + to display content
	Alert content has string length limits
	English: 19 characters
	English: 39 words

POW	Computes a specified power value based on a number. EX: (16 # 3: = POW,0.5); // 16^0.5 = 4 Note: Base must not be negative, otherwise the alarm will jump COR-122 POW(,) the first operand must not be negative
ROUND	Returns the value of a value after rounding the carry Ex: #10 := 1.4;

Function	Description
	#1 := ROUND(#10); // #1 = 1 #2 := ROUND(1.5); // #2 = 2

SIGN	Returns the positive and negative signs of a number, negative numbers are -1, positive numbers are 1,0 is 0
	Ex:
	#10 := 4;
	#1 := SIGN(#10); // #1 = 1
	#2 := SIGN(-4); // #2 = -1
	#3 := SIGN(0); // #3 = 0

SIN	Calculate the sine of a number
	Ex:
	10 := 90;
	#1 := SIN(#10); // #1 = 1
	#2 := SIN(-90); // #2 = -1

SLEEP	Temporarily give up the execution right of the macro loop, generally used to cooperate with the loop (FOR, WHILE...) Use, to avoid infinite circle use caused by man-machine card Ex: SLEEP();

SQRT	Calculate the square root of a number
	Ex:
	#10 := 4;
	#1 := SQRT(#10); // #1 = 2
	#2 := SQRT(9); // #2 = 3

STD(argument 1, argument 2)	Convert the value to an Input Unit (IU) based on parameter 17: Least Input Unit (LIU).
	Citation 1 is the value of the unit to be changed
	The argument 2 is a standardized unit, usually #1600, and the #1600 setting comes from the parameter 17 setting
	In the metric system:
	Ex1: When Pr17=2, #1600 corresponds to LIU = 0.001mm
	#9 := 100;
	#10 :=STD(#9,#1600); // #9 is 100 BLUs, so #10 is 0.1mm(100*0.001)
	Ex2: When Pr17=3, #1600 corresponds to LIU = 0.0001mm
	#9 := 100.
	#10 :=STD(#9,#1600); // #9 is 100*1000 BLUs, so #10 is 0.01mm(100*1000*0.0001)
	Under the British system:
	Ex3: When Pr17=2, #1600 corresponds to LIU = 0.0001inch
	#9 := 100;
	#10 :=STD(#9,#1600); // #9 is 100 BLUs, so #10 is 0.01inch(100*0.0001).

STDAX(arg	Convert the value into a standard unit corresponding to the axial

Function	Description
ument 1, argument 2)	direction The argument 1 is a number 2 is the name of the corresponding axis image Ex: #24 := STDAX(#24,X); #3 := STDAX(#3,A);

TAN	Calculate the tangent value of a number
	Ex:
	#10 := 45;
	#1 := TAN(#10); // #1 = 1
	#2 := TAN(-45); // #2 = -1

WAIT	The system stops presolving until WAIT has completed.
	For example:
	% @MACRO
	G10 L1000 P50 R1; // Fill in 1 for R50
	G94 G01 X100. F1; // Assume that the system is Reset
	WAIT();
	G10 L1000 P50 R0; // Fill in R50 with 0
	M30;
	Suppose you reset the system in the second row single-section move, since the third row exists
	WAIT relation, so the system stops presolving at the third line, so reset R50 content is still 1.

7.20 Compound fixed cycle（G70～G76）

Explanation:A fixed loop to simplify the program. In the finishing shape program, the path of rough cutting tool can be determined automatically.

G code	Name
G70	Finishing cycle
G71	Longitudinal roughing cycle
G72	End rough cutting cycle
G73	Profile cutting cycle
G74	Face cutting cycle
G75	Longitudinal cut-off cycle
G76	Compound thread cutting cycle

In the above functions, the turning material is generally cylindrical, and in the CNC turning processing of a certain contour and size, need multiple rough cutting and a fine cutting, therefore, in the CNC controller constructed a group of fixed turning cycle, to achieve in the rough cutting and fine cutting, automatically generate a series of tool paths to complete the contour turning.

Reduces the workload while writing programs. These instructions automatically generate tool paths to complete workpiece contour machining with straight, taper, and circular architectures.

7.20.1 Fine cycle(G70)

⚠PAY ATTENTION

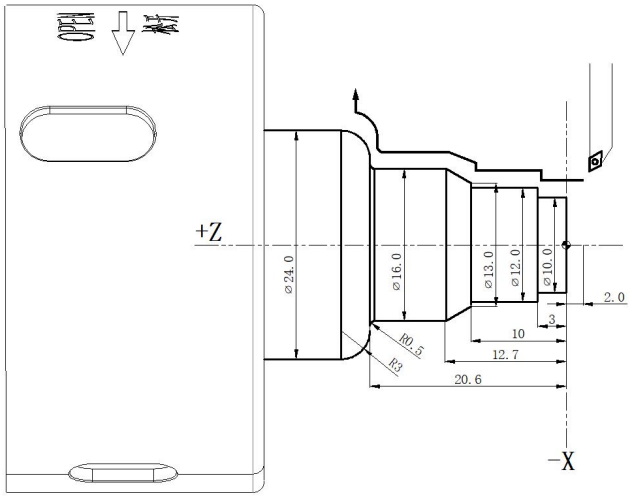
- F, S and T functions specified in single section G71, G72 and G73 are invalid, but F, S and T functions specified between serial number "ns" and "nf" in G70 are valid
- When the G70 cycle finishes, the tool goes back to the starting point and reads the next section.
- A sub-program cannot be called in any section between "ns" and "nf" used in G70 to G73.

Explanation: The G70 instruction is the finish turning cycle, also known as the contour turning cycle. This instruction must be used in conjunction with the block removal cutting cycle of the previous section. In general, the finish turning cycle is followed by the block cutting cycle in the program, and the execution range is only from the start section number "P (ns) "to the end section number "Q (nf)". After rough cutting for G71 / G72 / G73 cycle, fine cutting must be performed with instruction G70 to achieve the final required size.

Format:

G70 P Q ; (ns) (nf)

P	The starting section number of the cutting cycle
Q	End section number of a cutting cycle



Ex 1:

Program example: (G71 G70)
T0303 G18 M8 G99 M03 S1=1500 G0 X28. Z-2.00 G71 U2. R.2 G71 P100 Q110 U.4 W.2 F.1 N100 G0 X10. G1 Z3.0 F.05 X12. Z10.0 X13. X16. Z12.6 Z20.1 G3 X17. Z20.6 R.5 G1 X18. G2 X24. Z23.6 R3. N110 G1 X28. G0 Z-2.00 G70 P100 Q110 G0 Z-2.00 G0 X30. M01 . . .

7.20.2 Lateral (outside diameter) coarse cycle(G71)

Preface:

G71 instruction for transverse (outside diameter) rough turning circle, 扒 line after the workpiece can be processed to set to outline, reserved and set aside a certain amount as fine motor, the cutting cycle should be defined including the scope of the single section workpiece contour path, cutting depth of each rough turning and fine cutting depth reserve quantity and direction.

Format:

G71 U(Δd) R(e) ;

G71 P(ns) Q(nf) U(Δu) W(Δw) F S T;

Δd	Each cut depth along the X axis can be preset by system parameter #4013
e	Retracting amount, which can be specified as a default value by system parameter #4012
ns	Cycle start sequence number
nf	End of cycle sequence number
Δu	Reserve for finishing in the direction of X axis (OD)
Δw	
F	Reserve for finishing along the Z axis (length)
S	Feed rate
T	

Action description:

1. The tool should be quickly positioned (G00) to point A (starting point) before cycling.

2. Line 扒 G71 command, the cutter according to the set of fine car reserved quantity (for Δ U / 2 X axis and Z axis to Δ W) for its offsets, migration to point C.

3. After the tool moves Δd further along the X axis, it begins to feed to the profile.

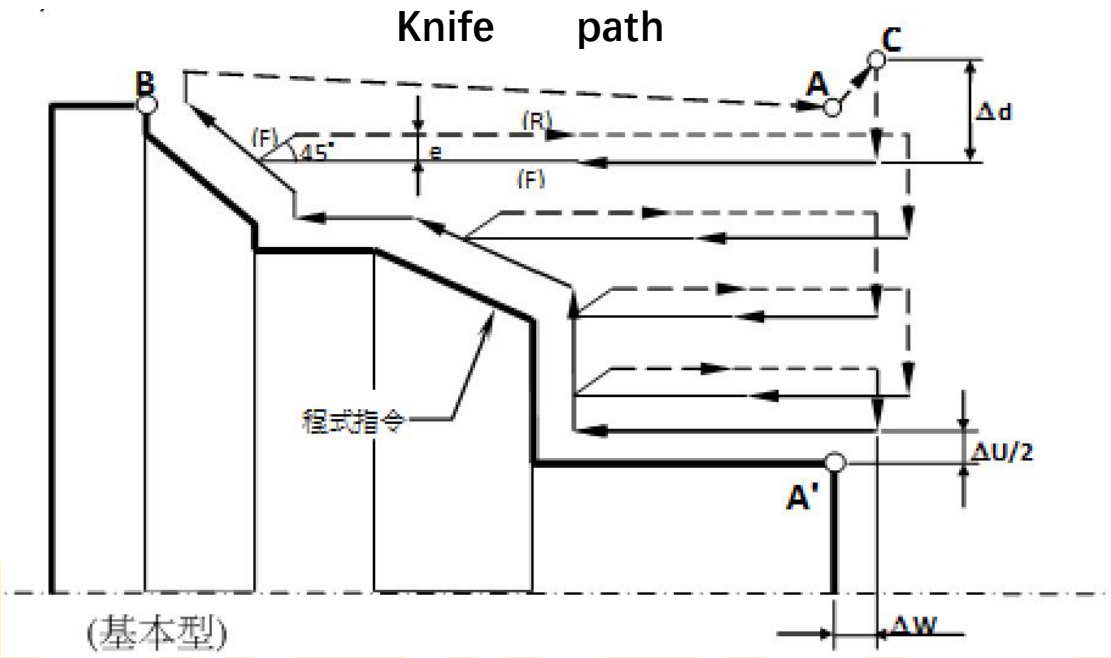
4. To 45 ◦ X axis direction after return e distance, Z axis feed in the opposite direction to the X axis parallel to the adjacent starting point.

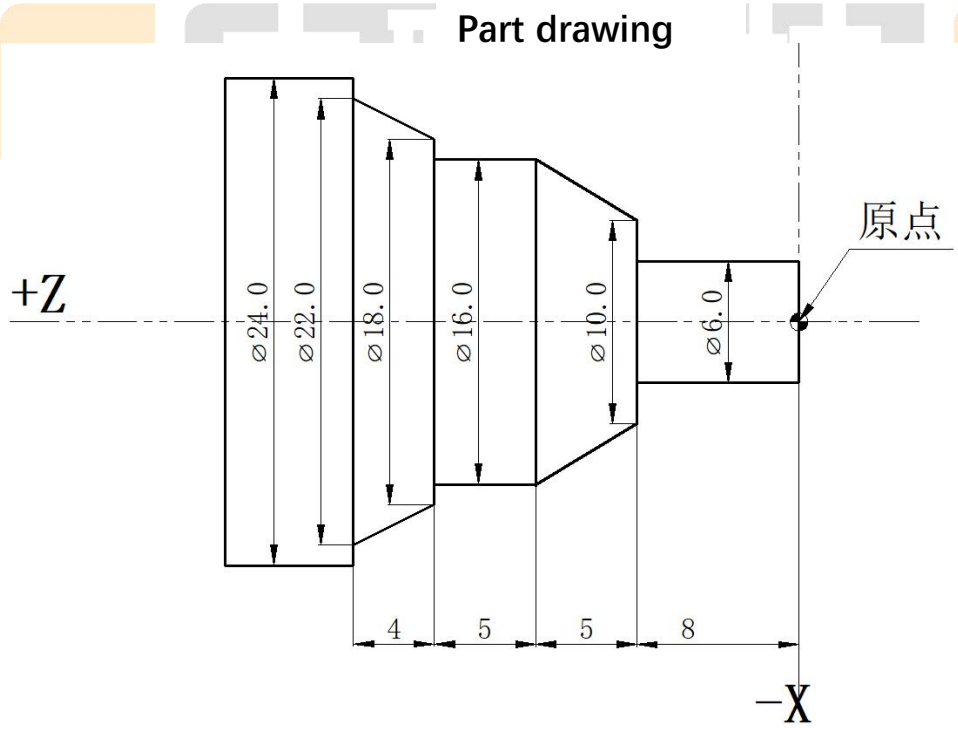
5. Then the X-axis moves Δd distance and the next cycle repeats.

6. At the end of the last cycle, the tool is turned along contour A and B once.

TYPE I: Usually used for processing from the end face, except that the first single section must conform to the condition that the X-axis is monotonically increasing (subtracting),

And each single section must conform to the condition that Z-axis is monotonically increasing (decreasing), and the next single section shall gradually increase (decreasing) compared with the last one.





Example program:(G71)
T0303 G18

```
M8
G99 M03 S1=1500
G0 X26.0 Z-2.
G71 U2. R1.
G71 P100 Q110 U.4 W.2 F.1
N100 G0 X6. 0
G1 Z8.0 F.1
X10.0
X16. 0 Z13.0
Z18.0
X18.0
X22.0 Z22.0
N110 X26.0
G0 X30.
Z-2.
M01
.
.
.
```

⚠PAY ATTENTION

- When ns and nf are not specified, U specified in G71 single section is the cutting depth ∆d, and vice versa is the reservation in the X-axis.
- The contour path is described by Blocks between ns and nf, from point A to point A' to point B. If the Z coordinates of the contour path are not increasing or decreasing in one direction, the system issues [MAR-002 turning path X, Z-axis only allows increasing or decreasing in one direction].
- The F, S, and T functions in the Block between NSNFS are invalid. These instructions are only valid if written in a single section of the rough turning cycle (G71).
- If the turning start point (the end point before G71) is lower than the contour path, the system issues [MAR-005 Turning start point is lower than the path]. Check the contour path to ensure that the starting point of turning is higher than the contour path.
- The individual sections between NSNFS cannot be used to call subroutines.
- Any instruction section contained in G71 will be invalid if it is corrected by the knife tip, but its complement value will be added to the reserved size.

7.20.3 Radial (end face) coarse cycle(Ｇ72)

Preface:

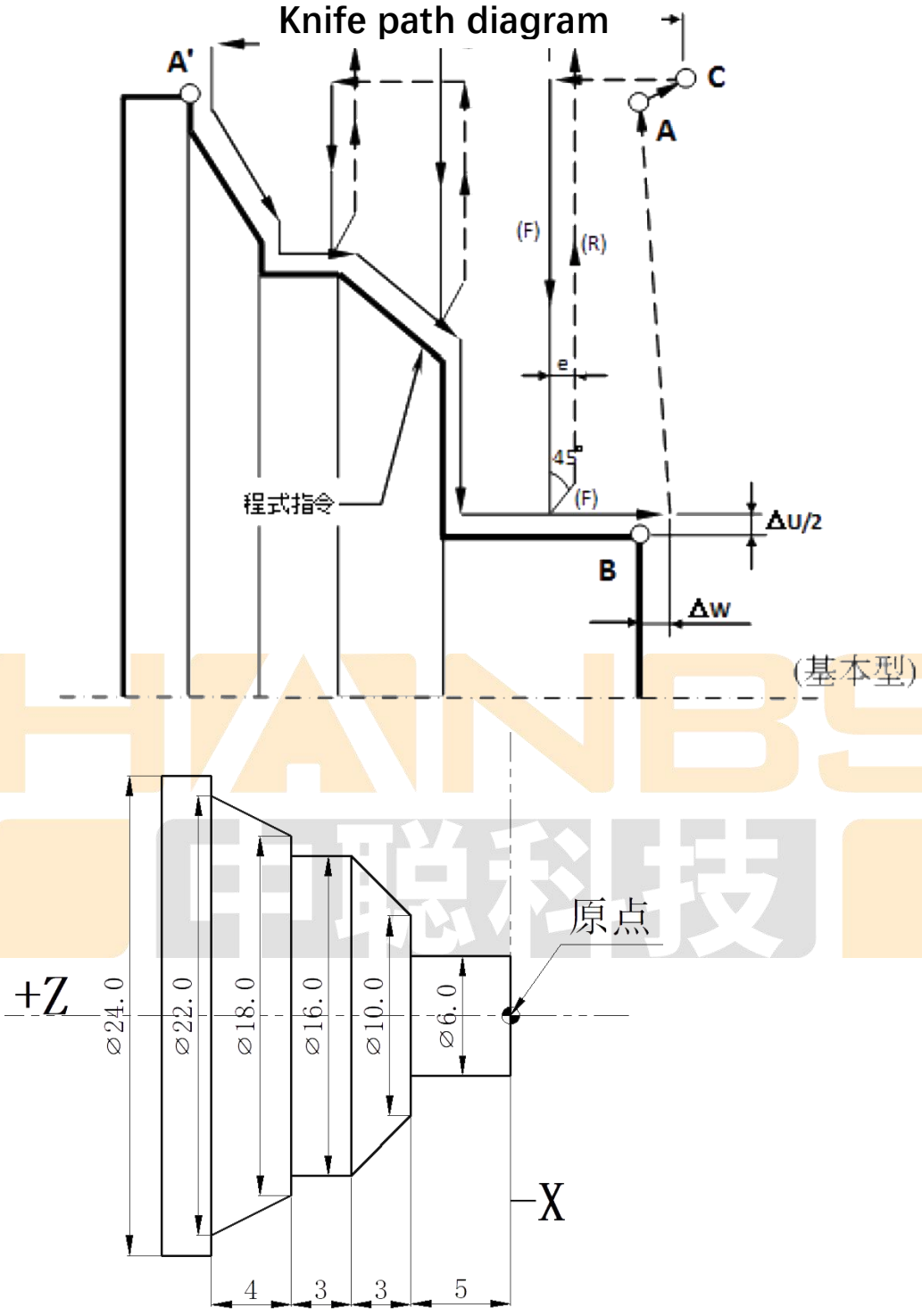
The G72 instruction is the radial (end) rough turning cycle. It is used to execute the turning process when the diameter of the workpiece is large and the length is short, that is, when the cutting quantity in the diameter direction is larger than the axial direction on the lathe

Format:

G72 W(d)　R(e)　;

G72 P(ns)　Q(nf)　U(Δu)　W(Δw)　F___ S___ T___;

d	Depth of each cut in z-axis direction
e	Retracting amount
ns	Cycle start sequence number
nf	End of cycle sequence number
Δu	Reserve for finishing in the direction of X axis (OD)
Δw	Reserve for finishing along the Z axis (length)
F	Feed rate
S	Spindle speed setting
T	Tool number



Example program:(G72)
T0303 G18 M8 M70

```
G99 M03 S1=2000
G0 X26.0 Z0. 0
G72 W2.0 R0.2
G72 P100 Q110 U0.4 W-0.2 F.1
N100 G0 Z15. 0
G1 X22.0 F.05
X18.0 Z11.0
X16.0
Z8.0
X10.0 Z5.0
X6.0
N110 Z0.0
G0 X26.0
Z-2.0
M01
.
.
.
```

⚠PAY ATTENTION

- The tool should be quickly positioned (G00) to point A (starting point) before cycling.
- Line 执 G72 command, the cutter according to the set of fine cutting reserve amount (for Δ U / 2 X axis and Z axis to Δ W) for its offset offset to point C,
- After the tool moves Δd further along the Z axis, it is fed to the contour surface.
- Then return to the Z axis with 45 ◦ direction e distance, the X axis feed in the opposite direction to the X axis parallel to the adjacent starting point.
- Then the Z axis moves Δd distance to continue the next cycle.
- At the end of the last cycle, the tool is turned along contour A and B once.
- After that, the cutter quickly locates to point A and waits for the next cycle to begin.

7.20.4 Forming contour rough turning (G73)

Preface:

G73 instruction for the shape contour rough turning cycle, with G73 (shape contour rough turning cycle), along the existing contour of the workpiece, repeat

cutting required times, each cycle move the appropriate distance, depth, repeat turning.

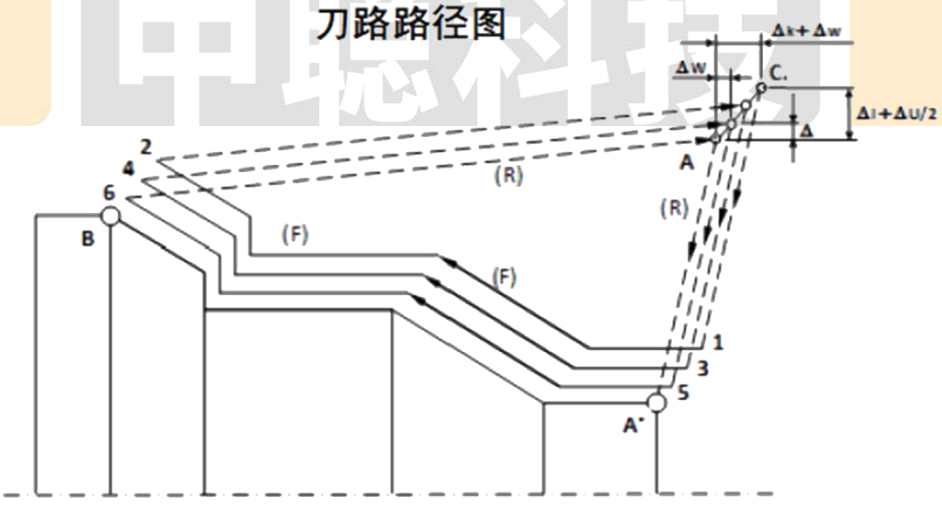
Format:

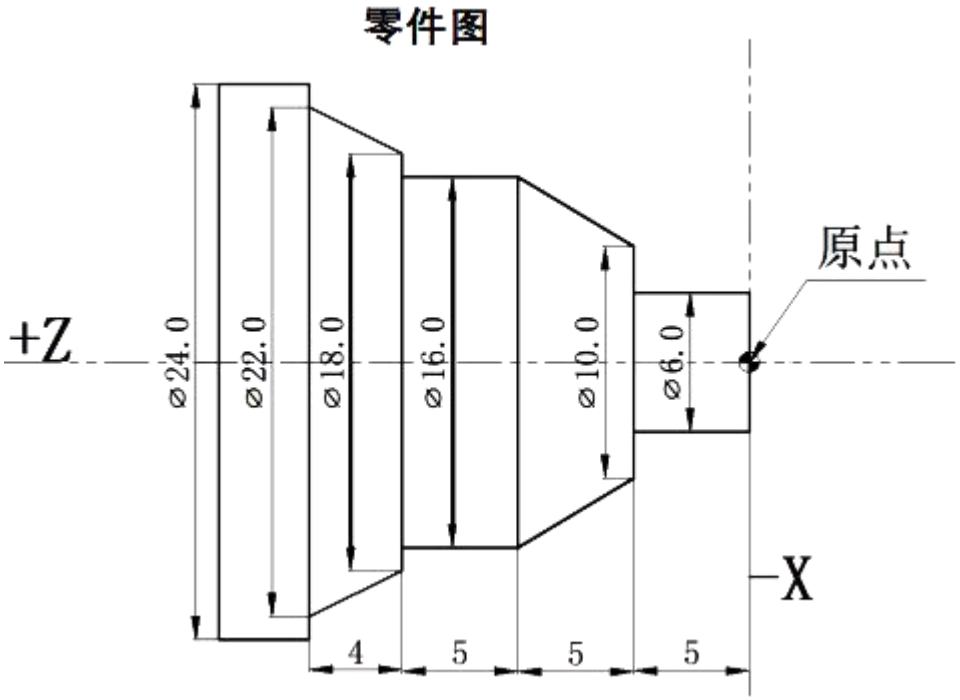
G73 U(Δi) W(Δk) R(d) ;

G73 P(ns) Q(nf) U(Δu) W(Δw) F___ S___ T___;

Δi	X direction (outside diameter) cutting quantity
Δk	Z direction (length) cutting amount
d	Number of cuts
ns	Cycle start sequence number
nf	End of cycle sequence number
Δu	Reserve for finishing in the direction of X axis (OD)
Δw	Reserve for finishing along the Z axis (length)
F	Feed rate
S	Spindle speed setting
T	Tool number

刀路路径图



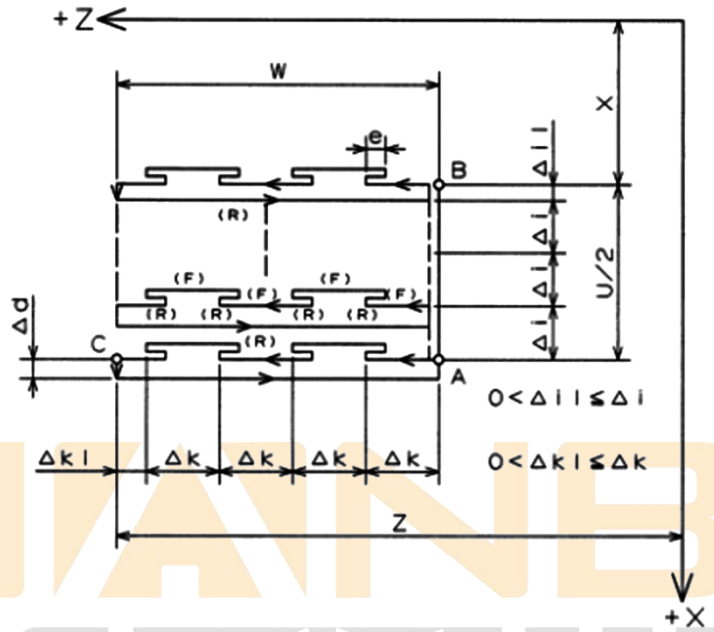


Example program:(G73)
T0303 G18 M70 G99 M03 S1=2000 G0 X26.0 Z-2.0 G73 U6.0 W-1.5 R3 G73 P100 Q110 U0.4 W-0.2 F0.1 N100 G0 X6. 0 G1 Z5.0 F0.1 X10.0 X16.0 Z10.0 Z15.0 X18.0 X22.0 Z19.0 N110 X24.0 G0 X28. 0 Z-2.0 M01 . .

7.20.5 Ending cutting cycle(G74)

Explanation: When the outside diameter cutting as shown in the figure is carried out, chip treatment can be effectively carried out.

In addition, when X(U) and P are omitted, only the Z-axis action is used for the deep hole drilling cycle.

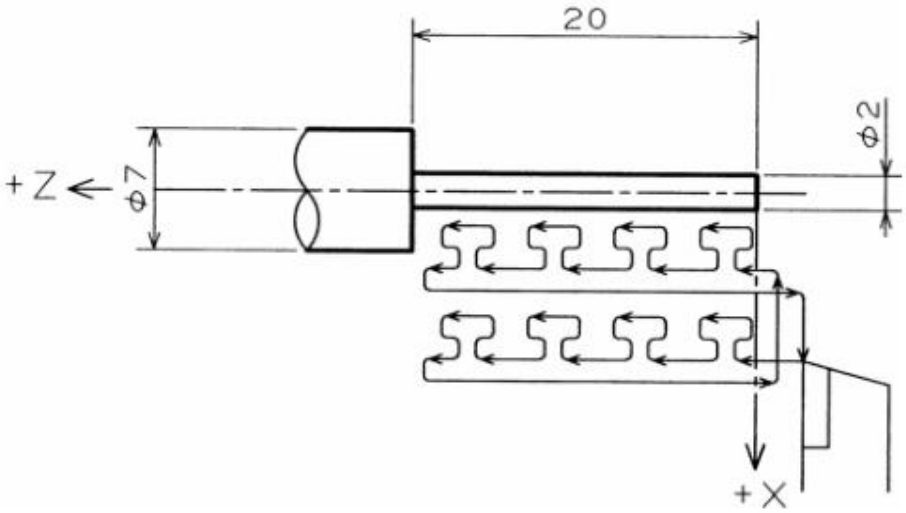


Format:

G74 R(e);

G74 X(U).... Z(W).... P(△i)....Q(△k).... R(△d).... F(f);

e	Return quantity This instruction is a modal instruction and is valid until the next instruction. It can also be set through the parameter (№8056), Change the parameter value according to the program instruction.
X	The X component at B
U	The projection from A to B
Z	The Z component of point C
W	The projection from A to C
△i	The amount of movement in the X direction (radius specified). Signed instruction.
△k	The entry in the Z direction. Signed instruction.
△d	Base cutting tool extension
f	Feed rate



Example program:(G74)
T0303 G18 M70 G99 M03 S1=2000 G0 X9.0 Z-2.0 G74 R0.5 G74 X2.0 Z20.0 P2.0 Q5.0 F0.1 G0 Z-2.0 X28.0 M01 . .

⚠PAY ATTENTION

- e and △d are used to specify parameter R. When there is lower X or Z, R represents the amount of X axis backcutting.
- There is only R parameter after G74, which means the amount of cutting in the direction of Z-axis. This is the mode G code. After one operation, the new program will disappear if it is always valid in this program.
- The value of Q is not set, the pecking action is cancelled, and the car reaches the terminal coordinate of Z-axis once.

7.20.6 Lateral (X-axis) peck cutting cycle （G75）

Preface:

G75 instruction is a horizontal (X-axis) pecking machining cycle. This instruction can be used for cutting ditches on the end face of the workpiece in X-axis and pecking drilling in X-axis.

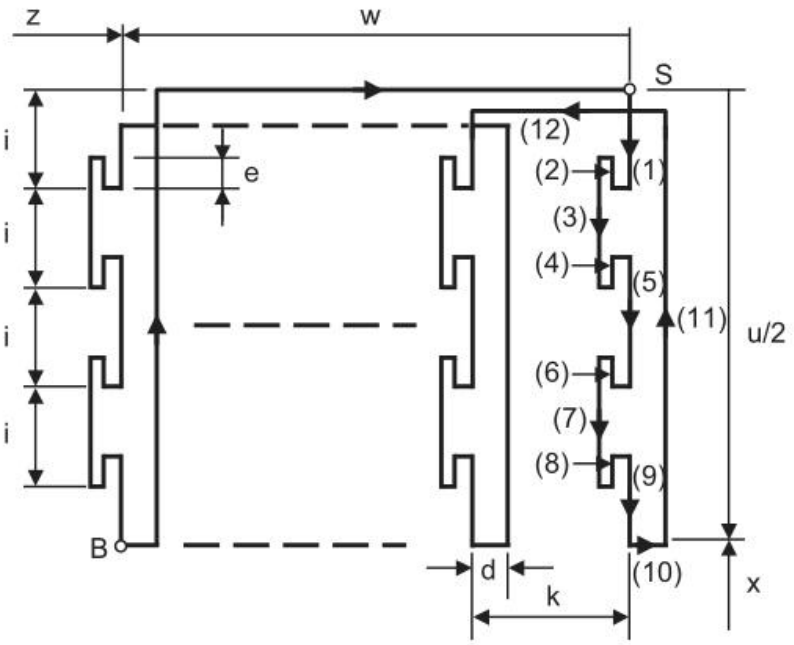
Such as cutting a groove on the outside diameter, in order to facilitate the screw back tool and avoid the terminal cutting out incomplete screw teeth, in addition to the lathe also often need to cut off the tool to cut off the workpiece processing, then need to use G75 instruction to complete.

Format:

G75 R(e)

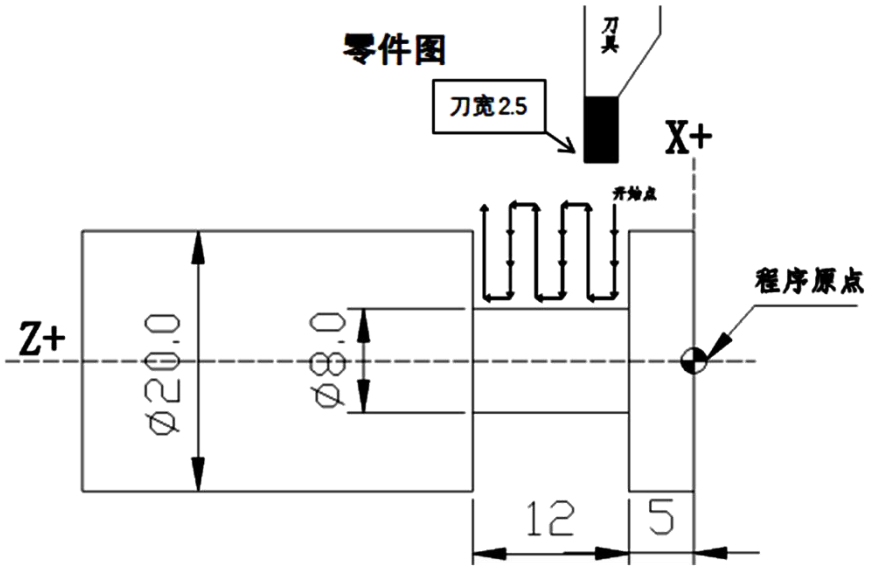
G75 X(U) Z(W) P(Δi) Q(Δk) R(Δd) F

e	Amount of cutting back (amount of cutting back after X axial cutting Δi)
X/U	X coordinates at point B (absolute/incremental value)
Z/W	Increment from point B to point Z (absolute value/increment value)
P i	Depth of cutting in each turn of X axis (expressed as radius value, positive value)
Q k	Z-axis travel length per cut (positive)
R d	To the end of the cutting, z-axis retreat (zero if the original path retreat)
F f	Feed rate



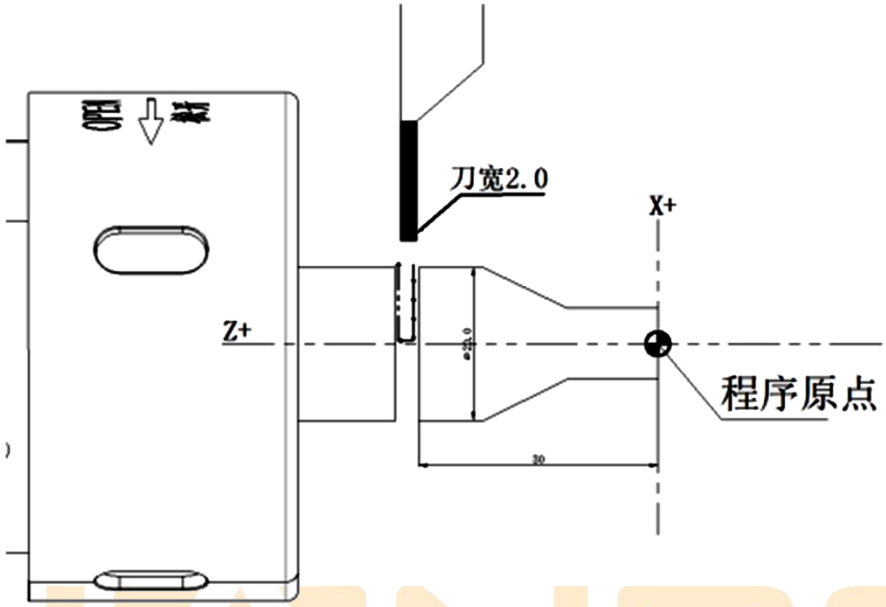
⚠PAY ATTENTION

- e and Δd are used to specify parameter R. When there is lower X or Z, R represents the amount of Z-axis backcutting.
- There is only R parameter after G75, indicating that it is the amount of cutting in the direction of X axis. This is the mode G code. After running it once, it is always valid in this program, and it will disappear in a new program.
- The value of Q is not set, and the pecking action is cancelled.



Slot cutting program example: (G75)
T0303 M08 G18 M70 G99 M03 S1=2000 G0 X22.0 Z7.5 G75 R0.5 G75 X8.0 Z17.0 P1.5 Q2.0 F0.1 G0 Z-2.0 X30.0 M01 . .

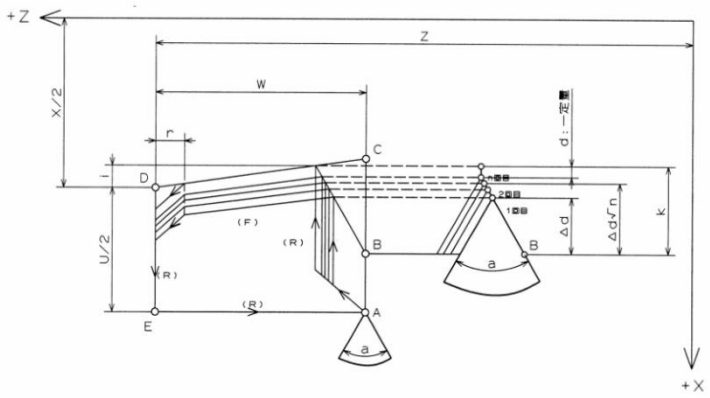
切断零件图



Cut-off procedure example:(G75)
T0303 M10 M08 M150 G18 M70 G99 M03 S1=2000 M104 S2=2000 G0 X22.0 Z32.0 G75 R0.5 G75 X-2.0 P1.5 F0.1 G01 X-2.0 F0.2 M05 G4X0.2 M11 . . M99

7.20.7 Compound thread cutting cycle （G76）

Explanation: Thread cutting cycle as shown below.



Format:

G76 P(m)(r)(a) Q(△dmin) R(d)；

G76 X(U) Z(W) R(i) P(k) Q(△d) F(L)；

m	The final finishing return number of 00 ~ 99 can also be set through the parameter № 4044. This instruction is a modal instruction and is valid until the next instruction.
r	Thread finishing (chamfer)00 ~ 99. Pitch L is input 0.1L in the range of 0.0L ~ 9.9L, which is a 2-digit 00 ~ 99 numerical instruction. This instruction is a modal instruction and is valid until the next instruction. This can also be set through the parameter № 4043. Change parameter values according to program instructions.
a	Angle of tool tip (Angle of thread tooth shape) The Angle instruction of 0° ~ 90° in 1° unit can be set through the parameter (№ 4042). This instruction is a modal instruction and is valid until the next instruction. m,r, and a are instructed at a time in the address character P. Example: m = 2, r = 1.2, a = 60 °, instructions for P021260.
△dmin	Minimum cut. When the first cut is less than (△d n-△d n-1) △dmin, the cut is constant △dmin. This can be set through the parameter № 4045. This instruction is a modal instruction and is valid until the next instruction.
d	Finishing allowance 0 ~ 9999μm. This can be set through the parameter № 4041. This instruction is a modal instruction and is valid until the next instruction.

i	The radius difference of the thread When i = 0, it is straight thread cutting.
k	Screw thread profile height (X axis distance command with radius  value).
△d	The amount of the first cut (radius specified).
F	Pitch of thread (metric thread lead).
E	Pitch of thread (inch thread lead).
H	The number of multiple snails

Example: Machining of M10*P1.5 thread length 14mm (including incomplete thread part with chamfer)

When the tool is installed on the tool holder fixed type 15mm offset [0G102]

〈program〉

```
G00 X12.0 Z13.0; ①
G76 P021060 Q0.05 R0.05 ; ②
G76 X7.83 Z30.0(W17.0) R0 P1.1 Q0.5 F1.5 H2; ②
```

※The Z-axis command value includes a tool tip offset of (15+1) mm

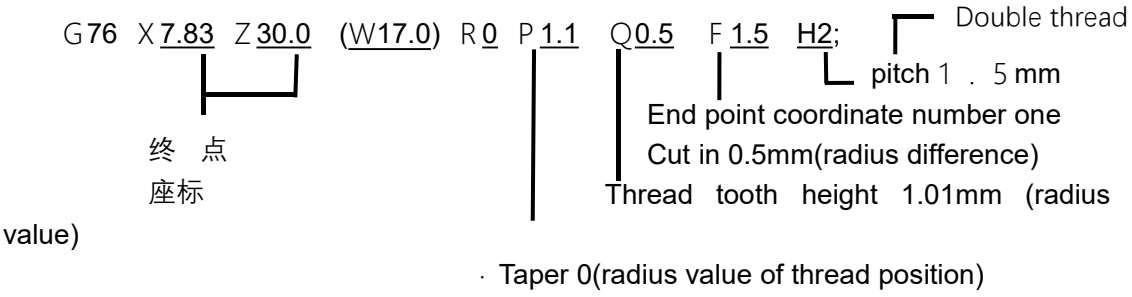
① : G0 Z; Please adjust the value according to the position of the tool tip.

In this example

Z = 16.0mm(tool tip position)-3.0mm(clearance) = 13.0.

②: The instructions are as follows.

```
                                  Minimum insertion 0.05mm(radius value)
                                Precision 0.05mm(radius value)
G76 P02   10   60   Q0.05  R0.05 ;
                    Tip Angle 60°
              Chamfer measure 1 pitch
        The final precision return times 2
```



Z = 16.0mm(tool tip position)+14.0mm(thread length) = 30.0

W = 17.0mm(tool movement in Z-axis during threading) = 14.0mm(thread length)+3.0mm(clearance)

⚠PAY ATTENTION

- The whole spindle multiplier lock is the multiplier set when entering the tooth cycle. That is to say, the multiplier twist control is ineffective in the tooth cycle until it leaves the tooth cycle.
- When the tool is removed, the ratio torsion control will be returned. Therefore, if the spindle ratio is not 100%, the spindle will increase and decelerate frequently.

7.21 Drilling fixed circulation（G80,G83～G87,G88)

A drilling program that simplifies a number of specific program segment actions into a single program segment.

⚠PAY ATTENTION

- Please command in Z-X plane (G18).
- Reverse the bit with M04 instruction.
- The presence or absence of Q arguments determines whether the G83/G87 action is a continuous or intermittent feed.

7.21.1 Canceling the drilling fixed circulation (G80)

Cancel the hole machining cycle with the G80 instruction (G83 ~ G85,G87,G88,G89) and perform the usual action afterwards.

7.21.2 Front drilling cycle(G83)

◇format

G83 Z(W)..... R.... Q.... P... F... ;

Z(W)	Format hole bottom coordinate instruction
R	The distance from the origin to point R is instructed in relative coordinates
Q	Each cut by radius (Q1.0 for 1mm)
P	Pause time at bottom of hole (instruction P1.0 when pause 1 second)
F	Cutting feed rate
K	Number of repeats

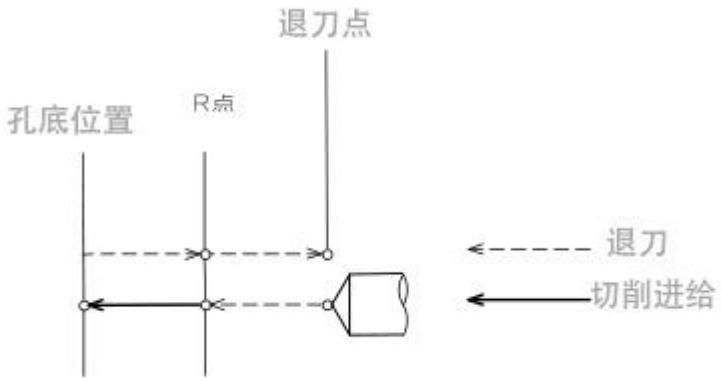
⚠PAY ATTENTION

- The corresponding pecking drill type is specified by user parameter Pr4001.
- When R point plane is set lower than the hole bottom plane, for example, R value is greater than the distance from the hole bottom plane to the initial point, the system sends out [MAR-011 drilling (pond) hole circulating feed plane R is lower than the hole bottom plane].
- If the G83/G87 command does not specify the hole bottom plane (Z/X coordinates), the system will issue the "MAR-012 drilling (pond) loop does not specify the hole bottom plane".
- Using G83/G87 command, such as each feed distance Q=0, the system issues [MAR-013 pecking drill cycle does not specify the feed quantity Q].

◇Action diagram

Intermittent feed when Q instruction is available. Without Q instruction, feed to hole bottom in one cut.

When there is no Q instruction



With Q instruction

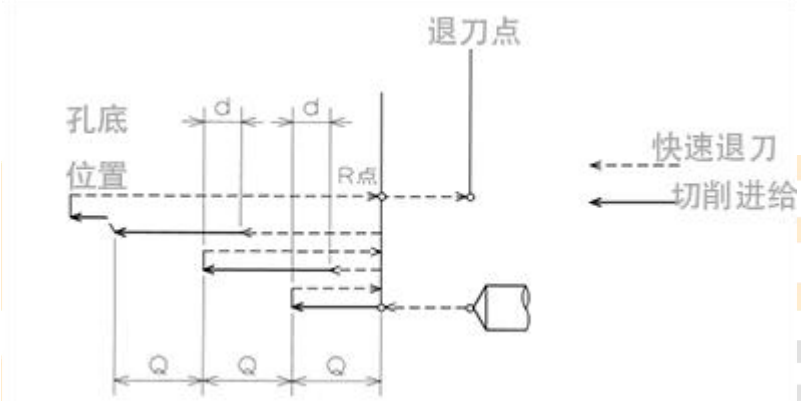
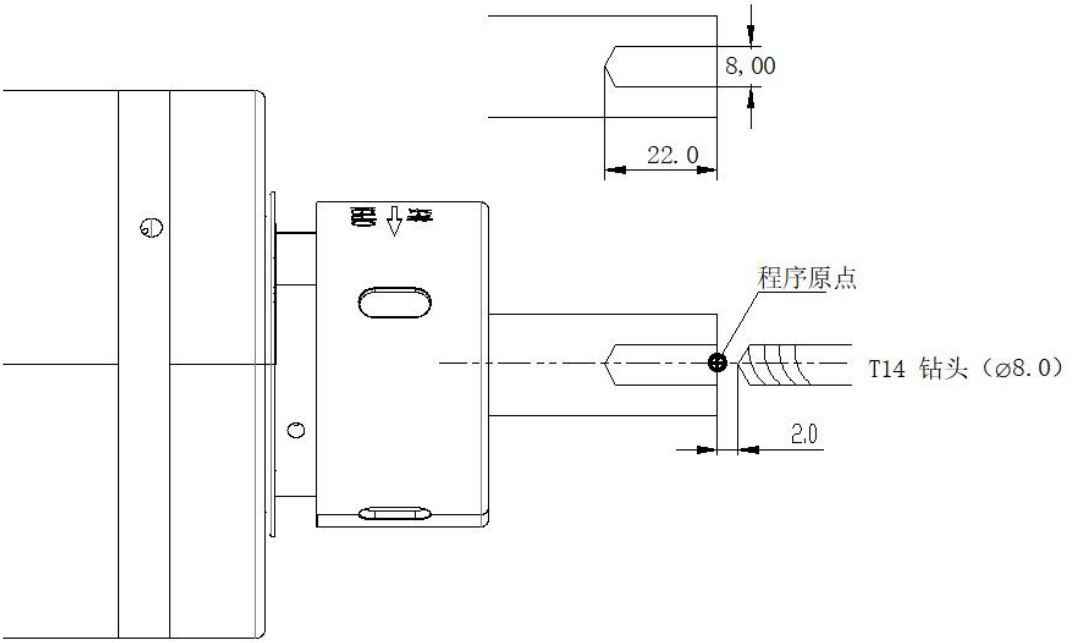


Illustration (drilling cycle)



Example program:(G83)
T1313 M70 M08 G99 M03 S1=2000 G0 X0.0 Z-2.0 G83 Z22.0 Q1.0 F0.1 G80 G00 Z-2.0 M01 . .

7.21.3 Positive forward tapping cycle(G84)

◇format

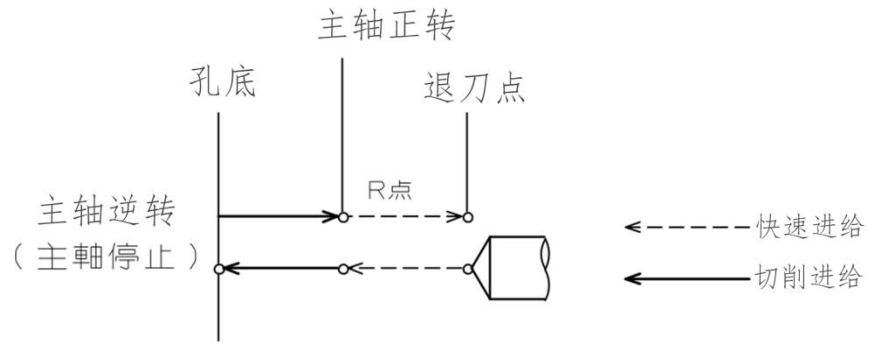
G84 Z(W).... R.... P....Q.....D.... F.... ;

Z(W)	The coordinate command of the hole bottom
R	Distance instruction from origin to point R in relative coordinates
P	Pause time at bottom of hole (instruction P1.0 when pause 1 second)
F	Cutting feed rate
Q	Specify how much to cut each time

⚠PAY ATTENTION

- The corresponding pecking pattern is specified by user parameter Pr4004.
- The spindle needs to run before tapping.
- At the end of G84, the spindle will return to the state before tapping. For example, before tapping, the spindle will turn positive, and after tapping, the spindle will automatically turn positive.
- G84 must be cancelled with G80, or if the program encounters G00, G01, G02, G03 or other cyclic G codes, this mode will also be cancelled automatically, otherwise it will persist (inheriting the status G code).
- Backhand teeth can be done by G84. By adding M4 behind G84, the backhand teeth can be processed.
- Before G84 tapping, the machining spindle (M70 first spindle, M71 third spindle) should be specified.
- During tapping, if the pause or reset key is pressed, the tapping action will be completed and stop at point R.

◇动作速 Degree diagram



Graphic (forward tapping)

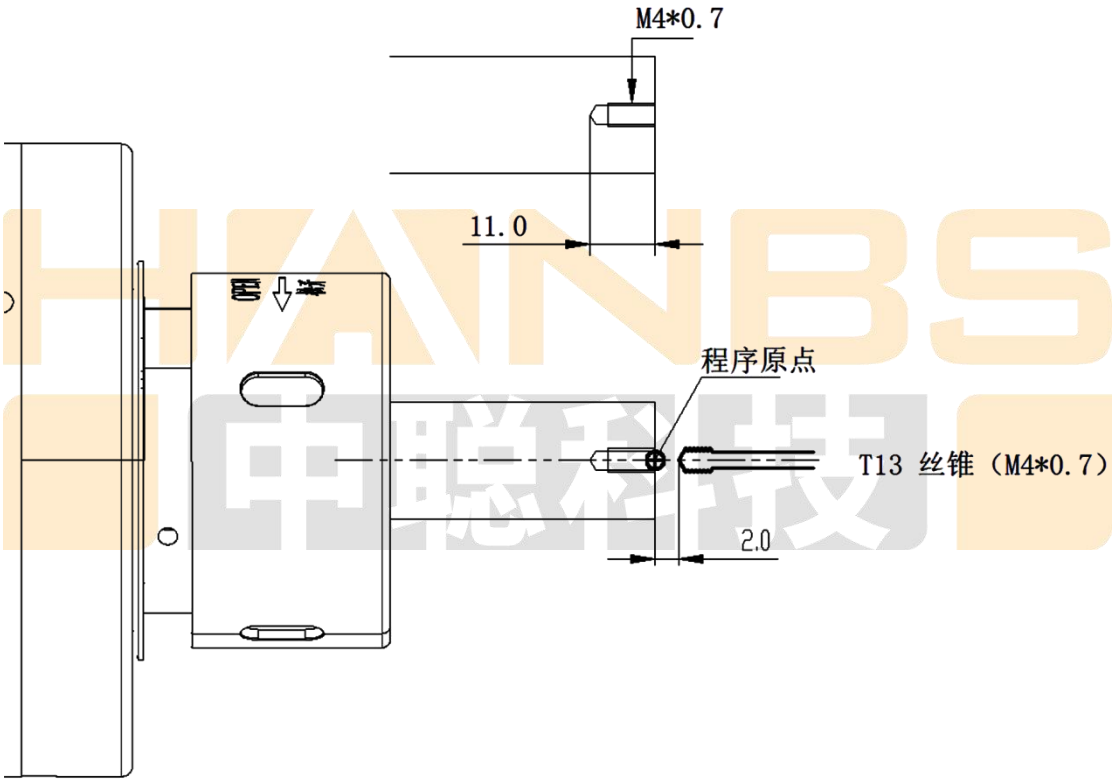


Illustration (face power tapping)

Orthodontic procedure example:(G84)
T1313 M05 M70 M08 G99 M03 S1=400 G4X0.5

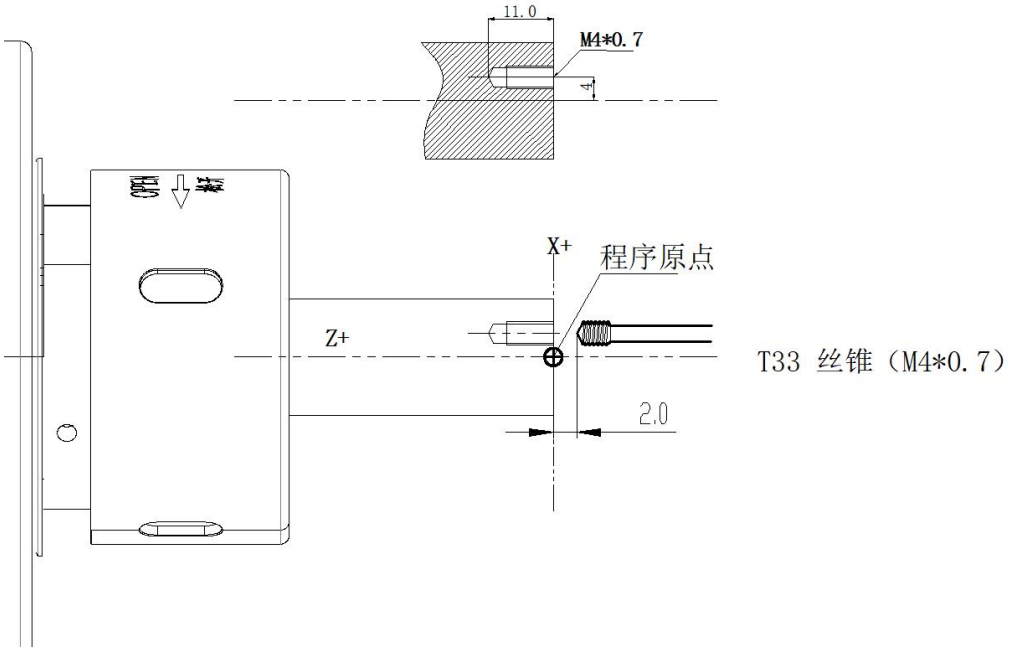
```
G0 X0.0 Z-2.0
G84 Z8.0 Q2.0 F0.7 M03
G80
G00 Z-2.0
M01
.
.
```

Reverse dental procedure example:(G84)

```
T1313
M05
M70
M08
G99 M04 S1=400
G4X0.5
G0 X0.0 Z-2.0
G84 Z8.0 Q2.0 F0.7 M04
G80
G00 Z-2.0
M01
.
.
```

Reverse dental procedure example: (G84)

```
T3333
M05
M71
M80
M08
G0X8.0 Z-8.0
G0 Y0.0 C0.0 Z-2.0
G99 M34 S3=400
G4X0.5
G84 Z9.0 Q2.0 F0.7 M34
G80
G00 Z-8.0
M70
M81
M35
M01
.
.
```



Orthodontic procedure example:(G84)
T3333 M05 M71 M80 M08 G0X8.0 Z-8.0 G0 Y0.0 C0.0 Z-2.0 G99 M33 S3=400 G4X0.5 G84 Z9.0 Q2.0 F0.7 M33 G80 G00 Z-8.0 M70 M81 M35 M01 . .

7.21.4 Side drilling circulation(G87)

◇format

G87 X.... R.... Q.... P.... F... .;

X	The coordinate command of the hole bottom
R	The distance from the origin to R
Q	Every time cut in (1mm instruction Q1.0)
P	Pause time at bottom of hole (instruction P1.0 when pause 1 second)
F	Cutting feed rate

◇Action diagram

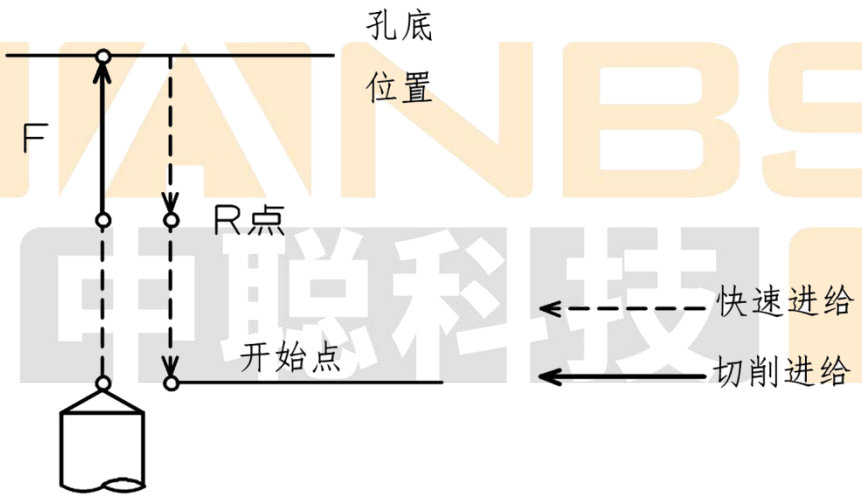
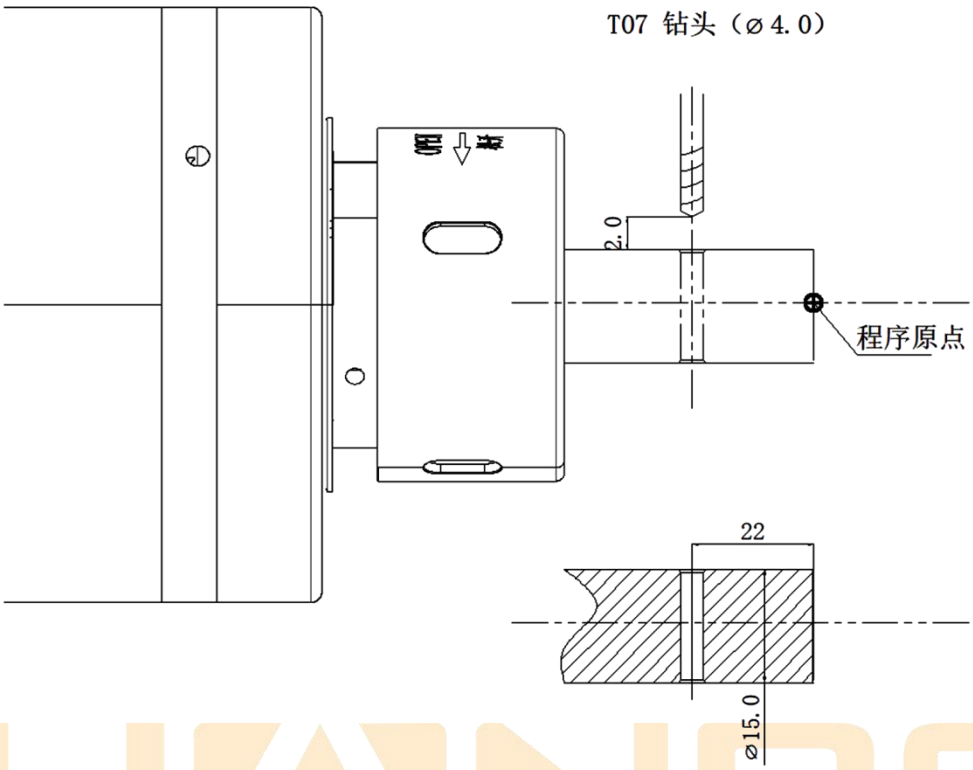


Figure (side drilling cycle)



Example program:(G87)
T0707
M05
M71
M80
G0 X30.0 Z22.0 Y0.0 C0.0
G98 M33 S3=3000
G4X0.2
G0 X17.0
G87 X-17.0 Q2.0 F150
G80
G00 X30.0
M81
M70
M35
M01
.
.

7.21.5 Forward sidel tapping cycle(G88)

◇format

G88 X.... R.... Q.... P....D... F... .;

X	The coordinate command of the hole bottom
R	The distance from the origin to R
Q	Every time cut in (1mm instruction Q1000)
P	Pause time at the bottom of the hole (P1000 when pause for 1 second)
F	Cutting feed rate

Figure (lateral forward tapping cycle)

Figure (lateral forward tapping cycle)

Figure (lateral forward tapping cycle)

⚠ PAY ATTENTION

- The corresponding pecking pattern is specified by user parameter Pr4004.
- The power head needs to run before tapping.
- At the end of G88, the power head will return to the state before tapping. For example, before tapping, the spindle will turn positive, and after tapping, the spindle will automatically turn positive.
- G88 must be cancelled with G80, or if the program encounters G00, G01, G02, G03 and other cyclic G codes, this mode will also be cancelled automatically, otherwise it will persist (inheriting state G codes).
- Backhand teeth can be done by G88. By adding M34 behind G88, the backhand teeth can be processed.
- Before entering G88 tapping, it is necessary to specify the machining spindle (M71 third spindle).
- During tapping, if the pause or reset key is pressed, the tapping action will be completed and stop at point R.

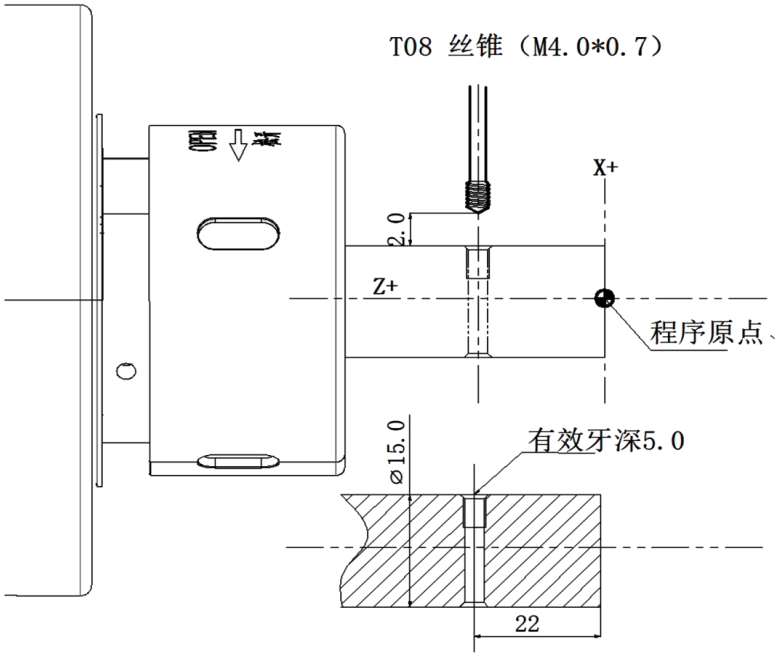


Figure (lateral forward tapping cycle)

Orthodontic procedure example:(G88)
T0808 M05 M71 M80 G0 X30.0 Z22.0 Y0.0 C0.0 G99 M33 S3=300 G4X0.5 G0 X17.0 G88 X4.0 Q2.0 F0.7 M33 G80 G00 X30.0 M81 M70 M35 M01 . .

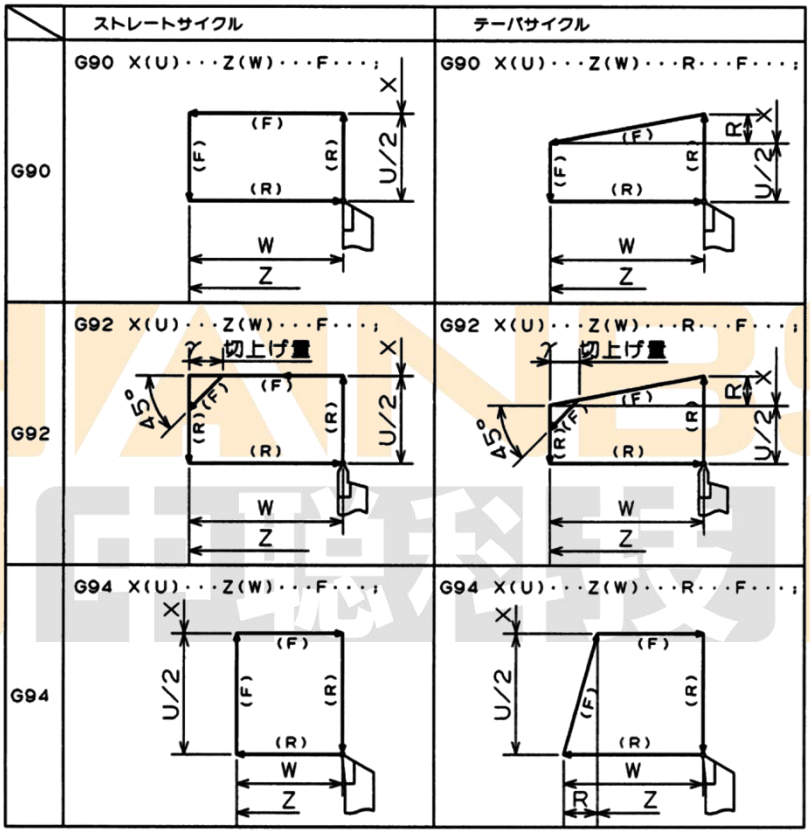
Reverse dental procedure example:(G88)
T0808 M05 M71 M80 G0 X30.0 Z22.0 Y0.0 C0.0 G99 M34 S3=300 G4X0.5 G0 X17.0 G88 X4.0 Q2.0 F0.7 M34 G80 G00 X30.0 M81 M70 M35 M01 . .

7.22 Single fixed cycle （G90,G92,G94）

Cutting action is equivalent to using G0,G1,G32 for "cut → cut (thread cutting) → reach → return" 4 program segments. Simplification of the above actions can be performed using this fixed loop.

■List of single fixed cycles

(R)···Fast forward speed, (F)···Cutting feed



7.22.1 Inside and outside diameter turning cycle(G90)

◇format

G90 X.... Z.... F... .;

G90 X.... Z.... R..... F... .;

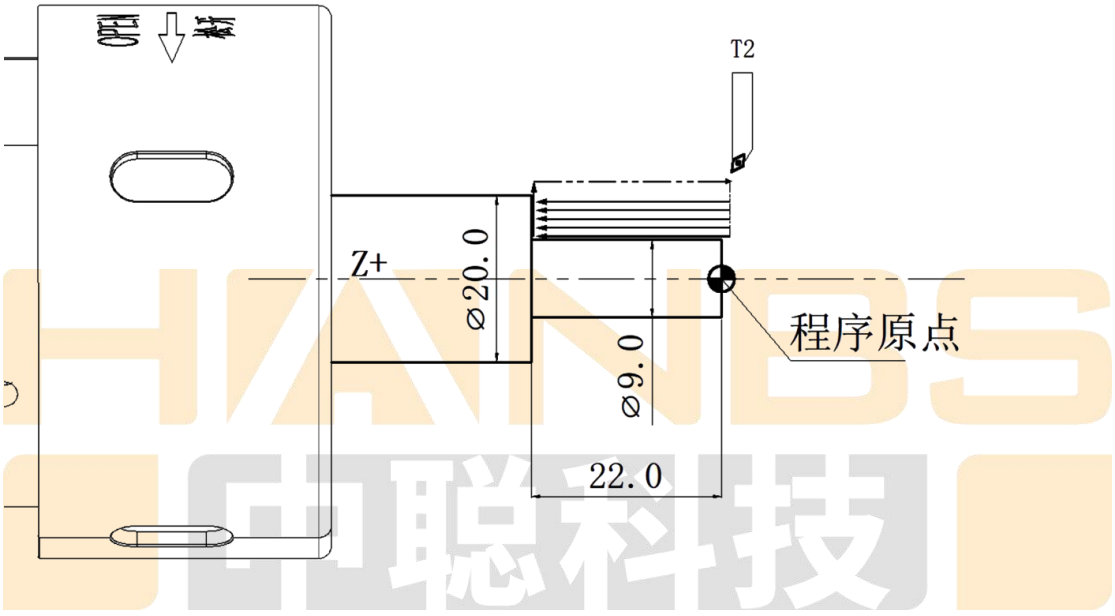
X	X axis turning end coordinates
Z	Z-axis turning end coordinates
R	The radius of the starting point and the ending point
F	Cutting feed rate

Reference:

G90 instruction is used to turn the axial cycle of outer (inner) diameter and taper. The function of the cycle is to turn the shape of several single section instructions. In the cycle instruction, only 1 single section can be used to simplify the processing program.

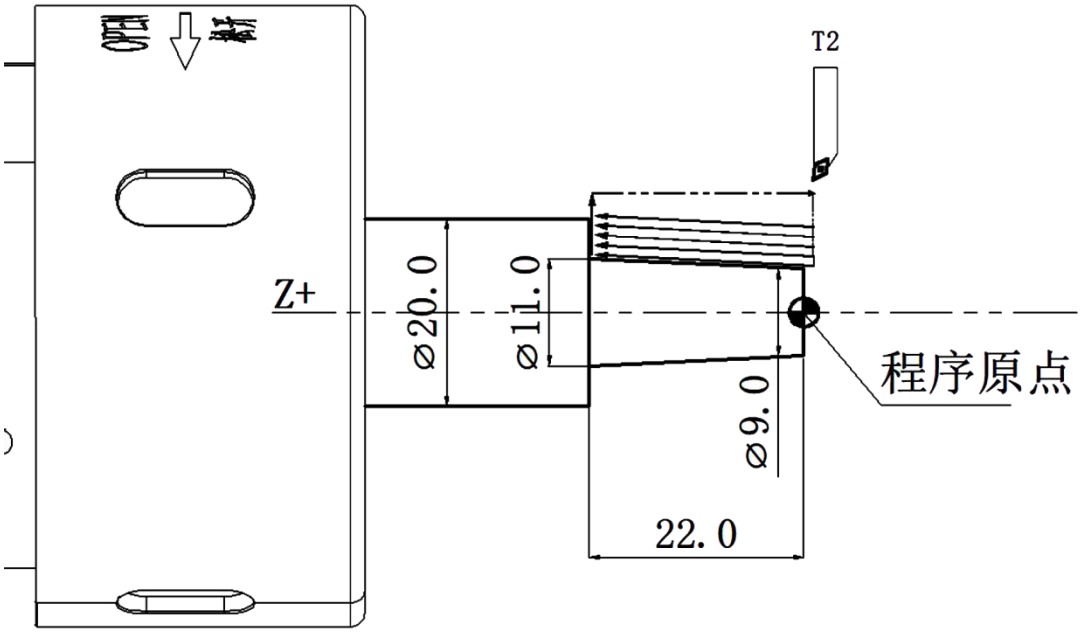
The symbol of the address symbol R is based on point B in the direction of point A '.

Figure (straight line turning cycle)



Example program:(G90)
T0202
M03 S1=2500 G99
M08
M70
G0 X30.0 Z-2.0
G0X21.0 Z-1.0
G90X19.0 Z22.0 F0.08
X17.0
X15.0
X13.0
X11.0
X9.0
G0X30.0
Z-2.0
M01
.
.

Figure (cone line turning cycle)

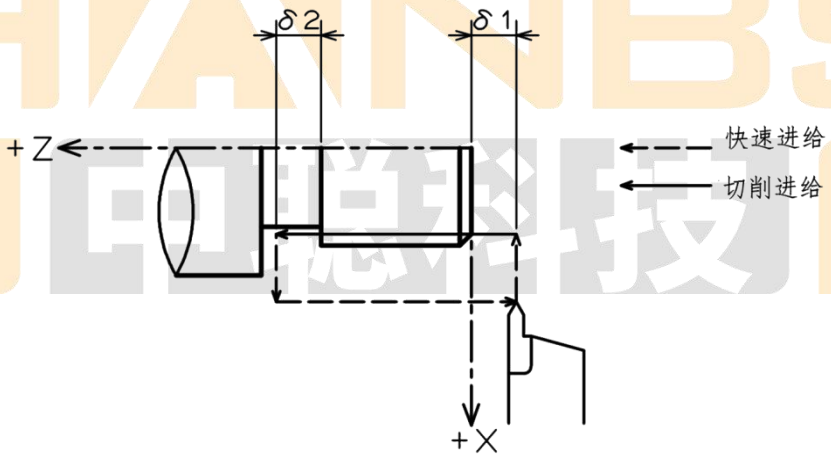


Example program:(G90)
T0202
M03 S1=2500 G99
M08
M70
G0 X30.0 Z-2.0
G0X22.0 Z-1.0
G90X21.0 Z22.0 R-1.0 F0.08
X19.0
X17.0
X15.0
X11.0
G0X30.0
Z-2.0
M01
.
.

7.22.2 Thread cutting cycle(G92)

⚠PAY ATTENTION

- Do not use tip R to correct.
- Incomplete pitch threads are produced at the beginning and end of thread cutting. Therefore, when determining the start and end of the cutting position, the following conditions need to be noted.
- Parameter Pr4018- thread turning tool chamfering Angle (θ) should be set in line with the actual turning tool Angle. For example, the actual turning tool Angle is 60 degrees. Parameter Pr4018 is set to 60 degrees.
- Handwheel simulation function is invalid with single section when turning teeth.
- If the feed position in the X direction is higher than the starting point, MARCO Alert 92 will be triggered to avoid interference between the tool and the workpiece.



The cut in the figure above begins with δ 1 and ends with δ 2 resulting in incomplete pitch threads.

Please calculate the values of δ1 and δ2 as follows.

$$\delta 1 = \frac{L \times S}{1800} \times 4 \qquad \delta 2 = \frac{L \times S}{1800}$$

L：螺纹螺距
S：轴回转速度

The speed of shaft rotation and the thread of the thread are restricted as next time.

L×S ≤ Maximum cutting feed speed (㎜ / min).

Straight thread cutting cycle

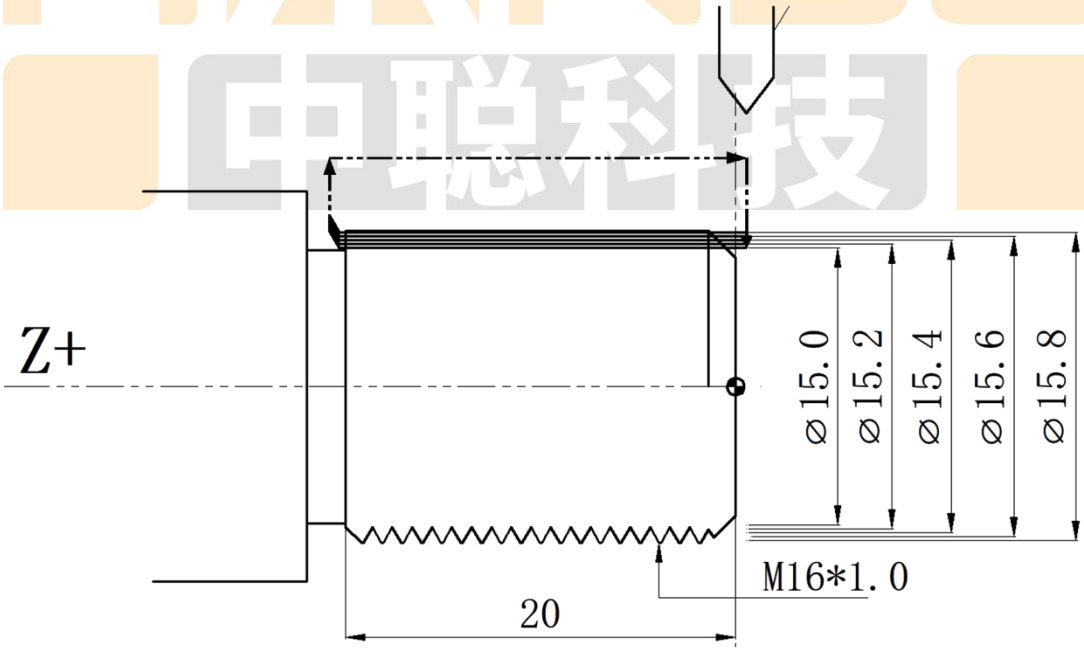
◇format

G92 X(U)···· Z(W)···· F ····H···· ;

X(U)	Terminal coordinates
Z(W)	
F	Pitch of thread
H	Command for multiple thread cutting, no command for single thread cutting.

Reference: The spindle ratio is locked at 100% when feeding the tool, and the spindle ratio is returned to torsion control when exiting the tool. Therefore, if the spindle ratio is not 100%, frequent spindle acceleration and deceleration will occur.

Figure (straight thread cycle)



Example program:(G92)
T0202 M03 S1=500 G99

```
M120
M08
M70
G0 X30.0 Z-2.0
G0X22.0
G92X15.8 Z21.0 F1.0
X15.6
X15.4
X15.2
X15.0
G0X30.0
Z-5.0
M01
.
.
```

Taper thread cutting cycle

◇format

G92 X(U)···· Z(W)···· R···· F···· H···· ;

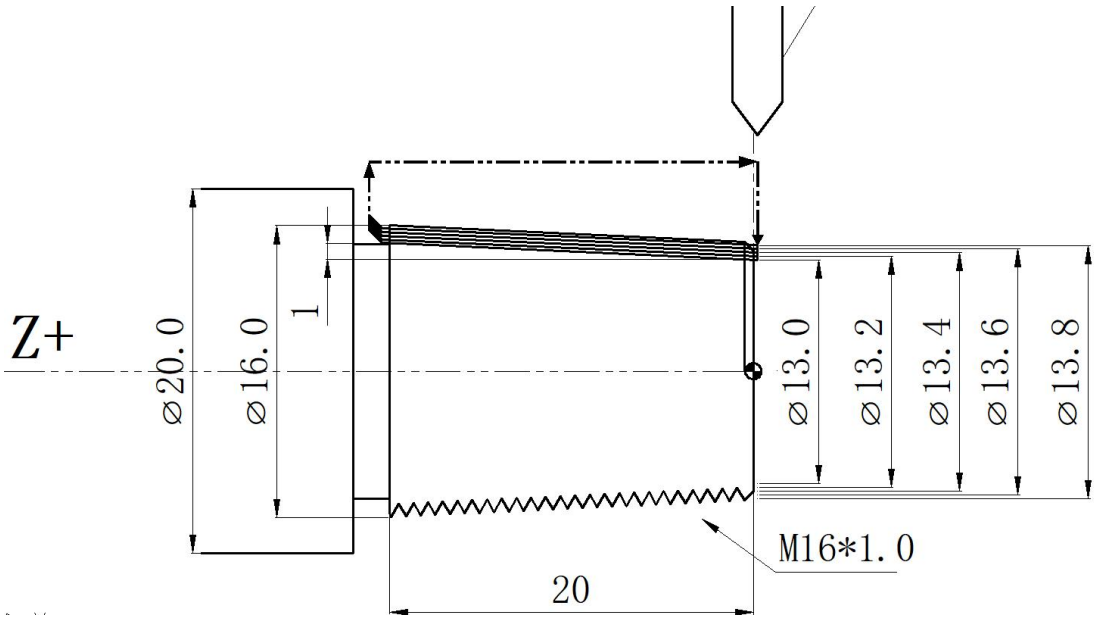
X(U)	Terminal coordinates
Z(W)	
R	X direction distance between point B and point A 'in the figure (difference of radius value)
F	Pitch of thread
H	Command for multiple thread cutting, no command for single thread cutting.

Reference:

X instruction is the absolute coordinate position instruction of point B.

The symbol of the address symbol R is based on point B in the direction of point A '.

Figure (cone thread cycle)



Example program:(G92)
T0202
M03 S1=500 G99
M120
M08
M70
G0 X22.0 Z-2.0
G92X15.8 Z21.0 R-1.0 F1.0
X15.6
X15.4
X15.2
X15.0
G0X30.0
Z-5.0
M01
.
.

7.22.3 End cutting cycle(G94)

Face cutting cycle

◇format

G94 X(U)···· Z(W)···· R···· F···· ;

X(U)	Terminal coordinates

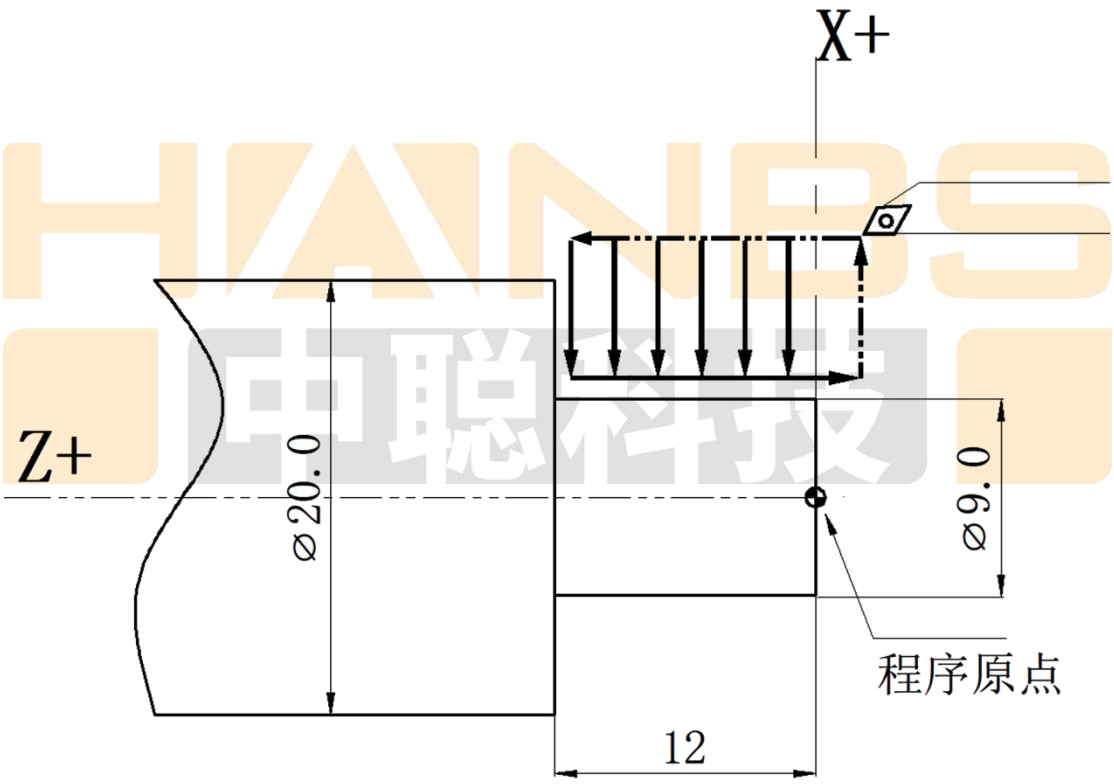
Z (W)	
R	The difference in length between the starting point and the ending point
F	Pitch of thread

Reference:

The G94 instruction is used for the end turning cycle, and the repetitive end turning of several single sections can be simplified into a single section.

Hand wheel simulation is effective with single section when face turning cycle.

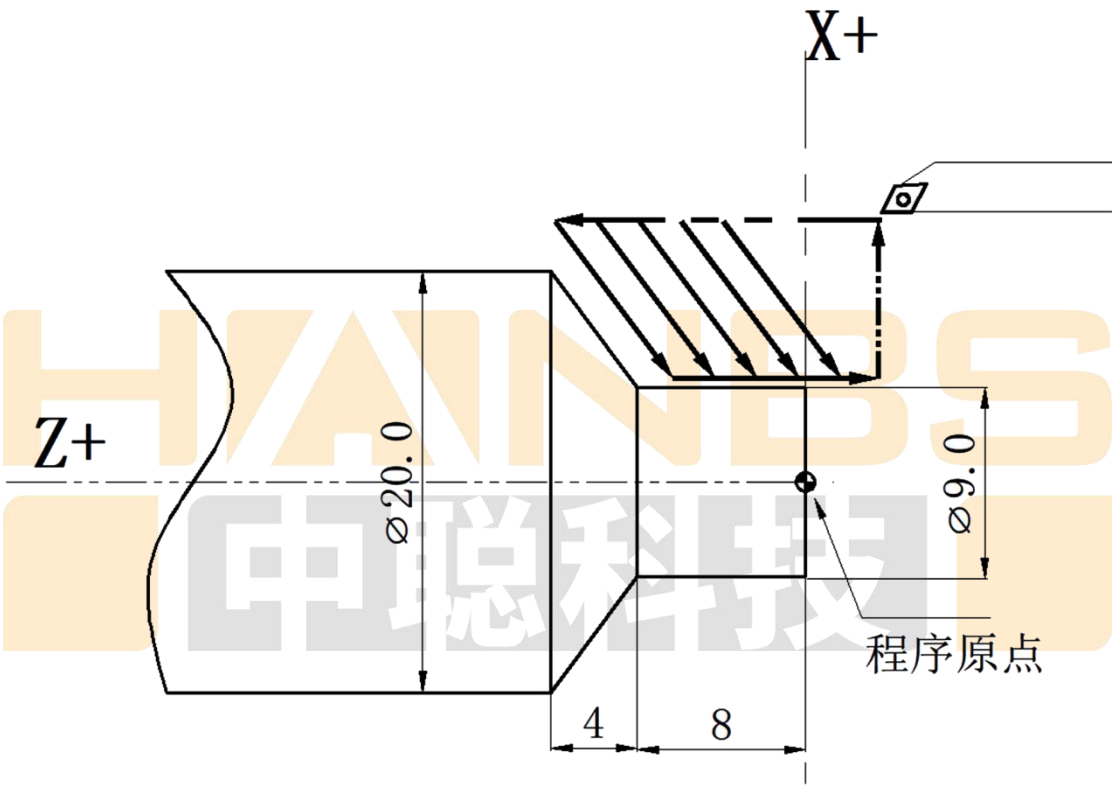
Graph (straight line loop)



Example program:(G94)
T0202 M03 S1=2500 G99 G4X0.3 M08 M70 G0 X22.0 Z-2.0 G94 X9.0 Z2.0 F0.1

```
Z4.0
Z6.0
Z8.0
Z10.0
Z12.0
G0X30.0
Z-5.0
M01
.
.
```

Figure (cone line cycle)



Example program:(G94)

```
T0202
M03 S1=2500 G99
G4X0.3
M08
M70
G0 X22.0 Z-5.0
G94 X9.0 Z-4.0 R4.0 F0.1
Z-2.0
Z0.0
Z2.0
```

```
Z4.0
Z6.0
Z8.0
G0X30.0
Z-5.0
M01
.
.
```

7.23 Cycle speed control（G96,G97）

Corresponding to the change of X axis position, with a certain speed to control the spindle rotation speed function.

7.23.1 Cycle speed must be controlled ON command

◇format

G96 S..... ;

S	Specifies the cycle speed (mm/min when input, ft/min when input, inch) Cycle speed control before G97 instruction.

7.23.2 Cycle speed must control OFF instruction

◇format

G97 S..... ;

S	Designation of turning speed (min-1) When there is no turning speed instruction, it is the turning speed of G96 instruction before G97 instruction.

7.23.3 Setting the maximum rotational speed of shaft

◇format

G50 S..... ;

S	Designation of turning speed (min-1) The maximum rotation speed of the spindle in the control of a certain cycle speed is given the rotation speed instruction in "S" after G50. Use this command to lock the maximum turning speed of the spindle.

⚠PAY ATTENTION

- In thread cutting engineering, do not use a certain speed control.

7.24 Feed function specification （G98,G99）

For cutting forward to F instruction to advance the unit (㎜ / rev) or to specify the switch function of the ㎜ / min.

7.24.1 Each minute feed(G98)

Instructions after the G98 instruction are executed in [mm/min] F instruction.

When the G98 instruction is performed at the $1 side, the X1 axis,Y1 axis and Z1 axis are fed in units of [㎜ / min].

When the G98 instruction is performed on the $2 side, the X2 axis and the Z2 axis are fed in mm/min.

Reference: Millimetre input unit is [mm/min].

The input unit of inch is [inch/min].

7.24.2 Each turn feed(G99)

All instructions after G99 execute F instruction in [㎜ / rev].

When the G99 instruction is performed on the $1 side, the rotation of the spindle corresponds to the X1 axis,Y1 axis and Z1 axis at the same time.

When the G99 instruction is performed on the $2 side, the auxiliary axis rotates at the same time corresponding to the X2 axis, and the Z2 axis is fed in unit at every turn.

Reference: Millimetre input unit is [mm/min].

The input unit of inch is [inch/min].

7.25 Main axis and phase synchronizations (G114.1), cancel synchronizations (G113)

Spindle synchronization control I specifies the synchronous spindle and the start/end of synchronization through the G instruction in the machining program.

7.25.1 The rotational speed is the same

Instruction format

G114.1 ;
G113 ;

Reference: If there are more than two spindles on the machine, the two spindles can be used to complete some special applications.
Angle phase is also the same or maintain a fixed Angle, and the two spindles to maintain synchronization in order to correct feeding.
Programming personnel in programming if do not need phase G114.1(R value can not be written).

Example program:1

Speed concurrent (cut off)			
$1 (main-axis program)		$2 (Back-axis program)	
T0101	Cutting knife	M105	Countershaft speed stops
G00X22.0 Z50.0	X and Z axes cut off position	T2020	The principal and auxiliary axes are concentric
M150	Chuck release allows rotation	M111	The countershaft chuck is released
M03 S1=2000	The main axis rotates clockwise		
M104 S2=2000	The counteraxis rotates counterclockwise		
G114.1	Rotational speed synchronization		
G4X1.0	Pause 1 second		
G04.1 P1	wait	G04.1 P1	wait
		G98 G0Z-140.0	Receiving material, fast displacement
		G1Z-150.0 F2000	Z-axis feeding position
		G4X0.1	Pause 0.1 second
		M110	Spindle clamping
G04.1 P2	wait	G04.1 P2	wait
G99 G1X-2.0 F0.03	Cut off		
G113	Cancel the same period		
G04.1 P3	wait	G04.1 P3	wait
M51	Test refuelling	G28W0.0	The Z2 axis quickly returns to the mechanical origin
M09	Cutting oil threshold	M105	Countershaft stop
M5	Spindle stop		
M11	Chuck release		

M97	Tool life reaches detection		
G04.1 P4	wait	G04.1 P4	wait
M99	End of program	M99	End of program

7.25.2 Phase synchronization

Instruction format

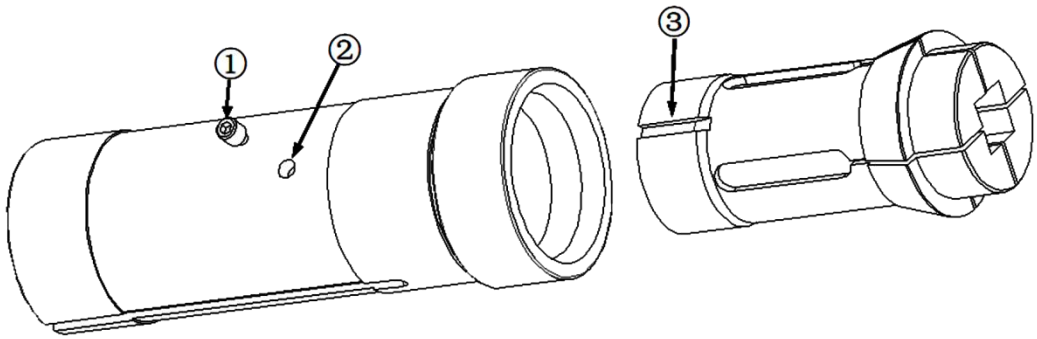
G114.1 R_ ;

G113 ;

R	Synchronous spindle phase offset

Reference: When the phase is the same, if the outside diameter of the back shaft clamping is round rod, the back shaft chuck can be fixed without pins.

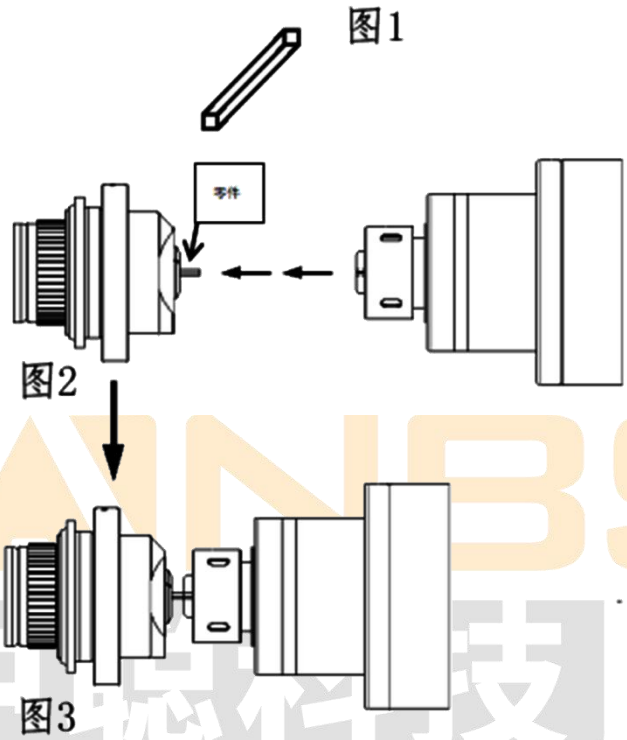
If the outside diameter of the back shaft is shaped, special attention should be paid to the chuck fixed with pin, to prevent the chuck position deviation caused by the collision when receiving material, before and after the chuck can be active.

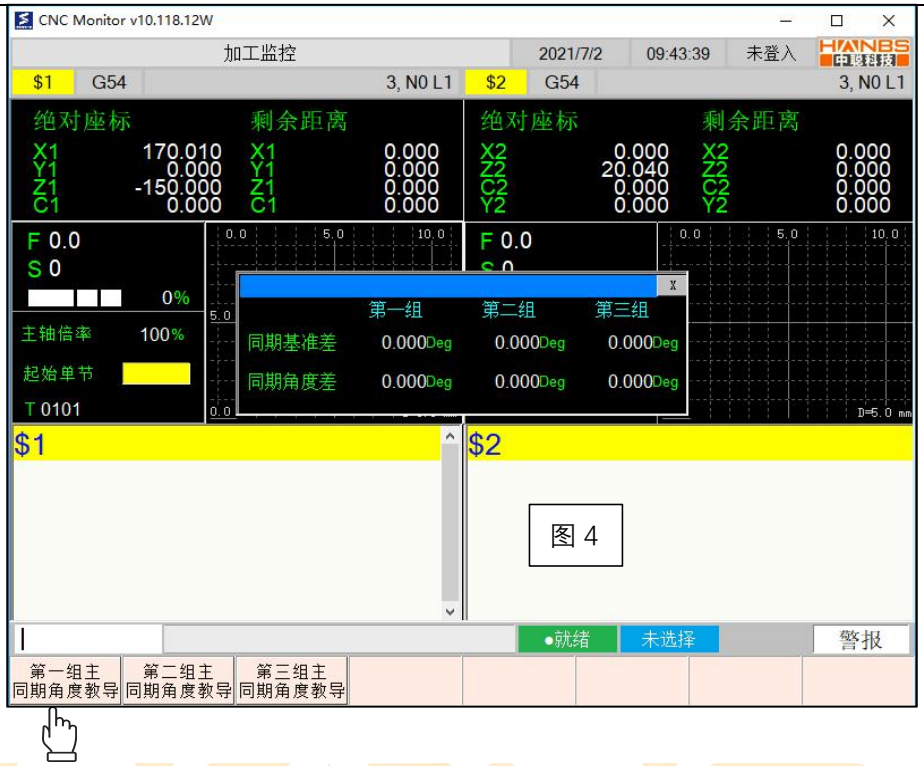


Operation	No	name	method
Fixed chuck	**1**	pin	Turn the pin clockwise to the ② position
	2	Pin hole	The pin should not protrude from the surface when mounted

Operation	No	name	method
	3	Chuck slot	The position of the card slot is aligned with the pin into the sliding sleeve, the chuck can move before and after

Adjust the phase of the main and back axes.





Operation	No	method
Phase adjustment	1	Quadrilateral materials as an example: Figure 1
	2	Put the material into the shaped guide sleeve, and the auxiliary shaft stays in the center of the main and auxiliary shaft: Figure 2
	3	With the C axis of the main and auxiliary shafts in free state, insert the chuck of the auxiliary shafts into the special-shaped parts: Figure 3
	4	The main and auxiliary shafts clamp the chuck.
	5	Find the "Simultaneous Angle Teaching" screen, and press "the first group of simultaneous Angle teaching" as shown in Figure 4.
	6	The auxiliary shaft releases the chuck and retreats to the origin of the Z axis

Program example: 1

Phase synchronization (cut off)			
$1 (Main-axis program 程序)		$2 (Back-axis program)	
T0101	Cutting knife	M105	Countershaft speed stops
G00X22.0 Z50.0	X and Z axes cut off	T2020	The principal and

	position		auxiliary axes are concentric
M150	Chuck release allows rotation	M111	The countershaft chuck is released
M03 S1=2000	The main axis rotates clockwise	;	
M104 S2=2000	The counteraxis rotates counterclockwise	;	
G114.1 R0.0	Phase synchronization	;	
G10 L1600 P1 Q100	Phase simultaneous completion	;	
G04.1 P1	wait	G04.1 P1	wait
;		G98 G0Z-140.0	Receiving material, fast displacement
;		G1Z-150.0 F2000	Z-axis feeding position
;		G4X0.1	Pause 0.1 second
;		M110	Spindle clamping
G04.1 P2	wait	G04.1 P2	wait
G99 G1X-2.0 F0.03	Cut off	;	
G113	Cancel the same period	;	
G04.1 P3	wait	G04.1 P3	wait
M51	Test refuelling	G28W0.0	The Z2 axis quickly returns to the mechanical origin
M09	Cutting oil threshold	M105	Countershaft stop
M5	Spindle stop	;	
M11	Chuck release	;	
M97	Tool life reaches detection	;	
G04.1 P4	wait	G04.1 P4	wait
M99	End of program	M99	End of program

7.26 C1,C2 concurrent period (M1000), cancel concurrent period (M1001)

Explanation: The axis movement instruction of the spindle (axis C1) overlaps, and the back spindle (axis C2) is moved.

Format:

M1000 ;

M1001 ;

Reference:

Do not carry out spindle speed command in C1-C2 axis concurrent control.

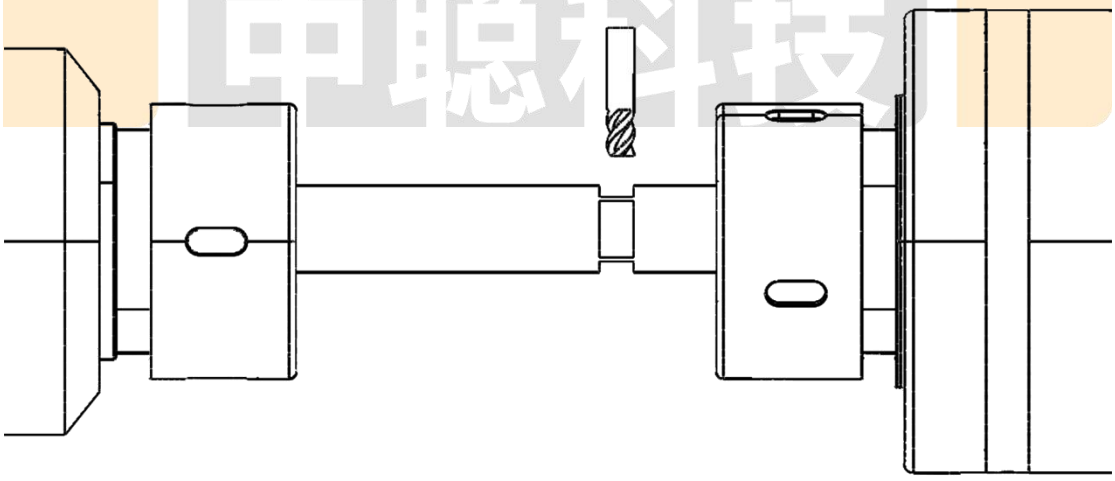
When using C1-C2 concurrent control, please stop the spindle before use.

Please note that the tool assembly interferes with the auxiliary shaft.

When C1-C2 axis concurrent instruction is used, the single system is invalid, and M1001 is valid only when $1 and $2 are executed simultaneously.

Generally C1 and C2 simultaneous functions are used together with Z1 and Z2 axis simultaneous functions

C-axis milling parallel diagram:



Example 2:

C-axis synchronization			
$1 (Main-axis program)		$2 (Back-axis program)	
		M105	Countershaft speed stops
G04.1P1	wait	G04.1P1	wait
M05	Spindle stop	;	
M80	C-axis ON	;	
M71	Specify the third spindle machining	G28 W0.0	Axis Z2 returns to the mechanical origin
T0707 （铣刀Ø8.0）	Milling cutter	T2020	Spindle concentric
M33 S3=2000 G98	Positive side power head turn	G0 X0.0	T20 X0.0
G0 X30.0 C0.0	X,C fast positioning	M111	The countershaft chuck is released
G0 Z34.0 Y18.0	Z,Y quick positioning	G4X0.1	Pause 0.1 second
G04.1P2	wait	G04.1P2	wait
;		G98 G0 Z-120.0	Z2 Quick positioning
;		G1 Z-150.0 F2000	Z2 Synchronize the clamping position
;		G4X0.1	Pause 0.1 second
;		M110	Auxiliary shaft clamping
G04.1P3	wait	G04.1P3	wait
M1000	C-axis is enabled at the same time	M1000	C-axis is enabled at the same time
G0 X22.0	X axis fast positioning	;	
G1 X15.0.0 F2000	Move to flat size	;	
G1 Y-18.0 F200	Milling the first flat bit	;	
G0 C90.0	C axis is 90°	;	
G1 Y18.0 F200	Milling the second flat bit	;	
G0 C180.0	C-axis is 180°	;	
G1 Y-18.0 F200	Milling the third flat bit	;	
G0 C270.0	C-axis is 270°	;	
G1 Y18.0 F200	Milling the fourth flat	;	

G0 X30.0	X axis tool lifting safe position	;	
M35	Power head speed stops	;	
M70	First spindle machining	;	
M81	C-axis OFF	;	
M1001	Axis C is closed at the same time	M1001	Axis C is closed at the same time
;		M111	Auxiliary shaft release
;		G4 X0.2	Pause 0.2 seconds
;		G28W0.0	Axis Z2 returns to the mechanical origin
G04.1P5	wait	G04.1P5	wait
T0 M01	Cancel knife fill and select Stop	T0 M01	Cancel knife fill and select Stop

7.27 Z1,Z2 concurrent (M2000), cancellation concurrent (M2001)

Explanation: By synchronous control, the movement of Z1 axis and Z2 axis can be synchronized, which can be used in the processing of special parts.

Format:

M2000

M2001

⚠PAY ATTENTION

- Usually the Z axis is used at the same time with the speed and the C axis at the same time.
- Remember to cancel the execution period after executing the same period.
- Programming personnel should fully consider the problem of axial travel and tool interference when programming, to prevent soft limit alarm, and collision machinery.
- M2000 and M2001 cannot be executed on a single system.
- The two systems must be valid when executing at the same time.

Part drawing:

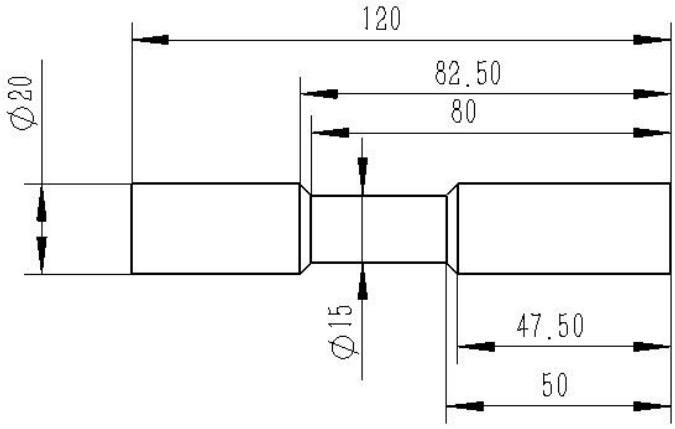
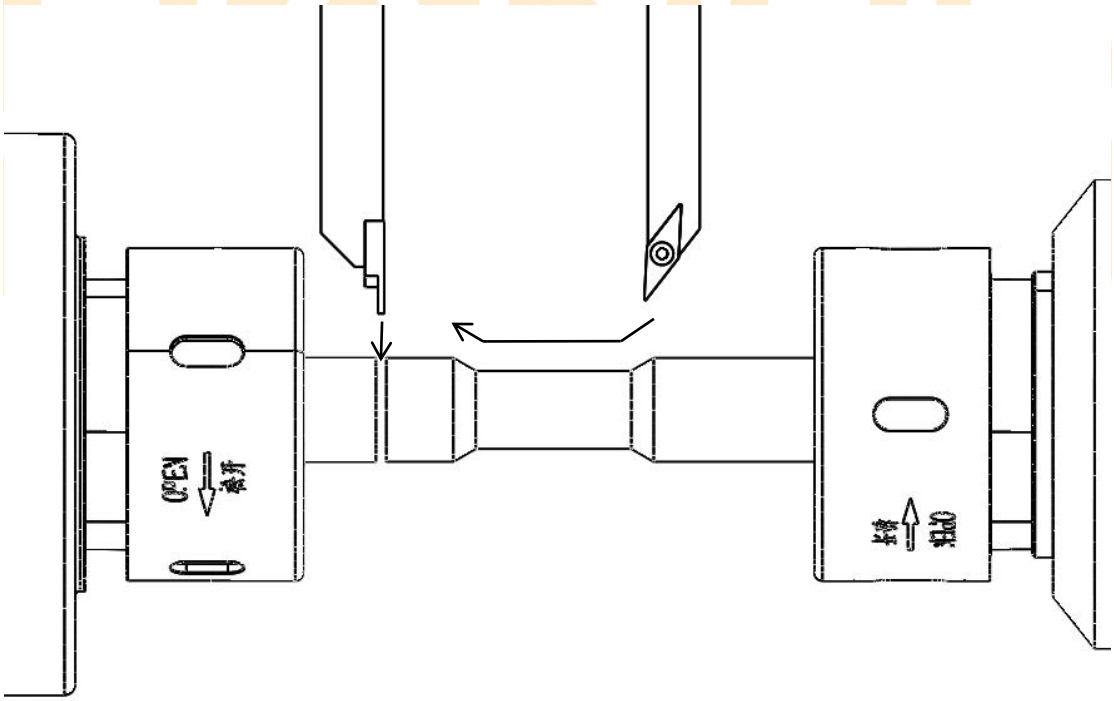


Diagram of turning period:



Example 1:

Z1,Z2Concurrent function + rotational speed concurrent	
$1 (Main-axis program)	$2 (Back-axis program)

T0303	T3 knife	G28 W0.0	Z2 axis back to mechanical position
M150	The chuck release allows the spindle to rotate	T2020	The principal and auxiliary axes are concentric
M03 S1=2500 G99	The main shaft is turning, feed every minute	G0 X0.0	T20 center point
M104 S2=2500	Counteraxis inversion	M111	The countershaft chuck is released
G114.1	Rotational speed synchronization	;	
G4X1.0	Pause 1 second	;	
G0 X30.0 Z47.5	X,Z quick positioning	;	
G04.1P1	wait	G04.1P1	wait
;		G98 G0 Z-120.0	Z2 feeding fast positioning
;		G1 Z-150.0 F2000	Z2 Synchronize the clamping position
;		G4 X0.2	Pause 0.2 seconds
;		M110	Auxiliary shaft chuck clamping
G04.1P2	wait	G04.1P2	wait
M2000	Z-axis opens at the same time	M2000	Z-axis opens at the same time
G0 X20.0	X axis fast positioning	;	
G1 X20.0 F0.1	Chamfering start point	;	
G1X15.0.0Z50.0F0.03	Working chamfer	;	
G1 Z80.0 F0.05	Outside diameter of vehicle	;	
G1X20.0Z82.5F0.03	Working chamfer	;	
G0 X30.0	Retracting tool	;	
T0 M01	Cancel knife fill and select Stop	;	
;		;	
T0101 （切刀宽度 2.0）	No. 1 cutter	;	
M03 S1=2200 G99	Positive spindle rotation	;	

G0 X30.0 Z122.0	X, Z axis fast positioning	;	
X22.0	X axis cutting point	;	
G1 X-2.0 F0.03	The X-axis cuts off the end point	;	
G113	Speed concurrent off	;	
M2001	The z-axis is closed at the same time	M2001	The z-axis is closed at the same time
G04.1P3	wait	G04.1P3	wait
M150	The chuck release allows the spindle to rotate	G28W0.0	
M51	Test refuelling	M105	
M151	Chuck release does not allow spindle rotation	G0W1.0	
M05	Spindle stop	;	
M09	Cutting oil threshold	;	
M11	Chuck release	;	
M97	Tool life reaches detection	;	
G04.1P4	wait	G04.1P4	wait
M99	End of program	M99	End of program

7.28 Setting coordinate system （G150）

■Coordinate system setting

◇format

G150 Z..... ;

<description>

. G150 is used to set the workpiece coordinate system, without the execution of G50Z, G50W instructions.

.Set G150 immediately after G299 and G300.

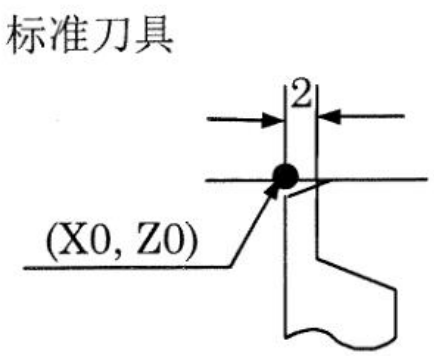
.At the end of the program, when the Z1 axis returns to the origin of processing, the instructions have been set with the coordinates of G150.

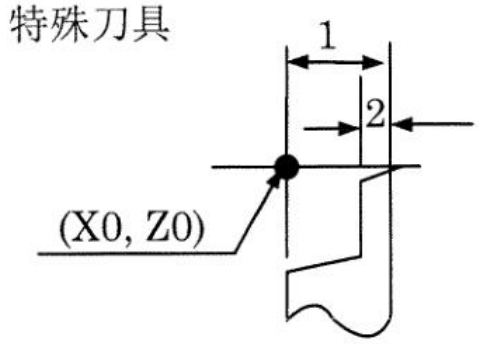
Program example G299 A10.0 X-2.0 Z20.0 T0101 M03 S1000 F0.03;

G300;
G150Z0.0 (Not processing end face);
G150Z-0.1 (End finish allowance is 0.1mm);

⚠PAY ATTENTION

- X axis value is set by G299, G150 does not need to be set.
- When adjusting the longitudinal deviation, it is best to take the benchmark after finishing the end face as the judgment standard.






```
G299A12.0 X-2.0 Z20.0 T0101 M03
S1500 F0.03;
G300;
G150Z-0.1;     (Forehand cutter)
M10;
.
.
.
.
M05;
M09;
M11;
G04.1P1;
M99
```

```
G299A12.0 X-2.0 Z20.0 T0101 M03
S1500 F0.03;
G300;
G150Z9.9;          (Backhand cutter)
M10;
.
.
.
.
M05;
M09;
M11;
G04.1P1;
M99
```

7.29 Processing intelligence settings（G299）

◇format

G299 A __X__ Z __M__S __F__ T__ ;

	content
A	Outer diameter of raw material
X	Cutter X axis end point
Z	Workpiece length + tool width
M	Cutting head rotation direction
S	Cut-off speed
F	Cut off the feed speed
T	Cutter number

⚠PAY ATTENTION

- When cutting the material, the machine will quickly move to A (raw material) +2mm and start cutting.
- In programming, the above processing parameters can not be less written or the machine will alarm.
- Please input with decimal point when programming.
- Before cutting head, make sure the tool is in a safe position to prevent collision.
- Cutting head, please manually cut the "hand wheel simulated opening".

7.30 Automatic cut-off（G300）

◇format

G299 A __X__ Z __M__S __F__ T__ ;

G300 ;

G150Z-0.1;

⚠PAY ATTENTION

- The G300 automatically cuts off the G299 processing intelligence input for use with the G150.
- Please process information setup instruction (G299) at the beginning of the $1 program.
- Make the cut-off end of the G299 instruction Xpos at cut-off time of the instruction in the program. consistent
- When the workpiece coordinate position of axis X1 (including wear correction) > the cut-off end of instruction G299, the cut-off action is carried out.

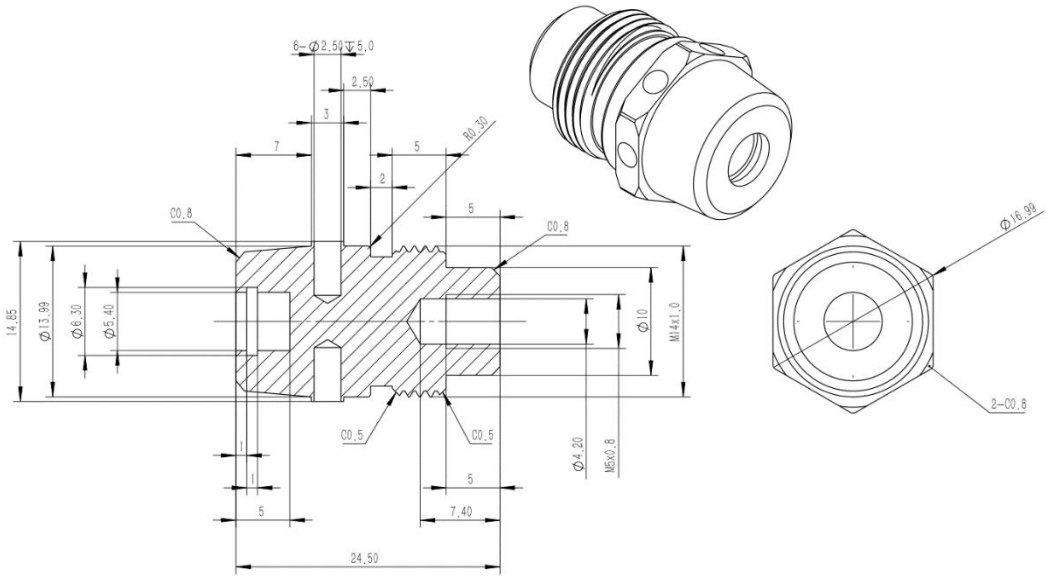
$1 (Main-axis program head)	
G299A...X...Z...M...S...F...T...	Processing information setting
G300	Automatic cutting
G150Z-0.1	Machining coordinate system setting
M10	The spindle chuck is clamped
G4X0.2	Pause 0.2 seconds
G0X30.0	Safe position of lifting knife
G0Z-4.0	Z-axis retreats to a safe position
M01	Selective stop
T0	Cancel cutting

Chapter Ⅷ Programming examples

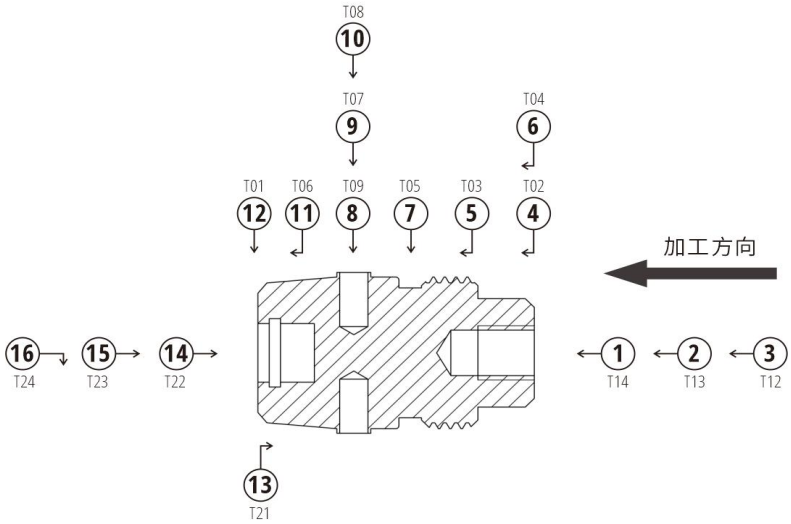
8.1 Processing configuration

This chapter explains the programming method according to the product below.

Raw material: SUS303 Φ17.0 bar.

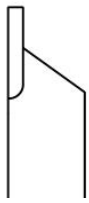
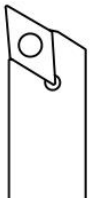
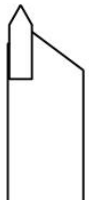
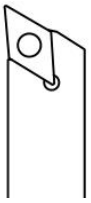
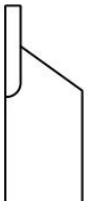
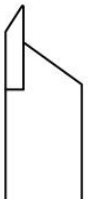


The processing sequence is as follows:

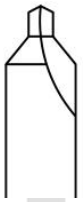


1. Center hole machining, 2. Drilling machining, 3. Tapping machining, 4. Rough outside diameter, 5. Turning teeth processing, 6. Finishing outside diameter, 7. Grooving processing, 8. Milling processing, 9. Side center hole machining, 10. Side drilling, 11. Back Outside Diameter Machining, 12. Cutting machining, 13. Back outer diameter machining, 14. Back drilling machining, 15. Back hole processing, 16. Back groove processing.

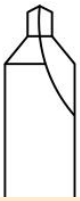
Turning tool

T01	T02	T03	T04	T05	T06
					
Cutting knife	Outside diameter turning tool	牙刀	Outside diameter turning tool	Slotted knife	Outside diameter turning tool
H2.0	R0.4		R0.2	H2.0	

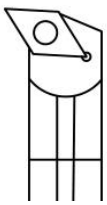
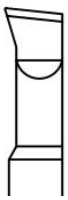
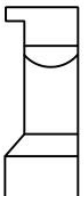
Milling unit cutter

T07	T08	T09	T10
			
Drill bit	Spot drill	Milling cutter	
Φ2.5			

Fixed drilling tool

T11	T12	T13	T14
			
	tap	Drill bit	Spot drill
	M5x0.8	Φ4.2	

Back tool

T21	T22	T23	T24
			
Outside diameter turning tool	Drill bit	Inner hole turning tool	Slotted cutter
	Φ5.2		

8.2 Preparation for processing

Explanation:

System program header description:

$1	**$1**	**$2**	
G299 A17.0 X-2.0 Z27.0 M03 S1500 F0.035 T0101	Processing information setting;	G28 W0.0	Z2 to zero
G300	Automatic cutting	M105	The auxiliary axis rotation stops
G150Z-0.1	End finish allowance 0.1	M110	Auxiliary shaft chuck clamping
M10	The spindle chuck is clamped	;	
G4X0.2	Pause 0.2 seconds	;	
G0X30.0	X axis back tool to safe position	;	
G0Z-4.0	Z-axis back tool to safe position	;	
M08	Cutting oil opening	;	
G04.1P1	wait	G04.1P1	wait
M01	Selective stop	M01	Selective stop

8.3 Center hole

Explanation:

The purpose of center hole machining is to make drilling more stable.

Procedure:

$1	
N14	Serial number (Cutter 14, same below)
T1414	Select the tool (T14 tool, 14 tool fill, the same below)
M3 S1=2000	Main axis 2000 turns positive
G0 G99 X0.0	Quickly locate to X0.0,
G0 Z-1.0	Quick positioning to Z-1.0
G1 Z0.5 F0.06	Cut feed to Z0.5
G0 Z-0.5	Back to Z-0.5
G1 Z2.85 F0.05	Cut feed to Z2.85

G4 U0.1	Pause 0.1 second
G0 Z-2.0	Back knife to Z-2.0
G0T0	Cancel cutting
M01	Selective stop

8.4 Drilling

Explanation:

Drilling and center hole machining methods are basically the same. However, when the machining depth is more than 2-3 times the diameter of the hole, it usually leads to blockage due to chip problems. In this case, it is necessary to withdraw the bit temporarily, remove the chips and then process again, which is called subsection machining. For deeper holes, internally cooled bits with intermediate cutting oil can also be used.

For segmented machining, please refer to the following table to control the feed amount:

material	cutter	First feed quantity	Subsequent feed amount
brass	Flat drill	Full length	-
Free-cutting steel	Twist drill	Triple diameter	0.6 times the first time -0.9 times the diameter
Tool steel, carbon steel, easy cutting stainless steel	Twist drill	2.5x diameter	0.5 times the first time -0.9 times the diameter
Stainless steel, hard to cut material	Twist drill	Double diameter	0.4 times the first time -0.9 times the diameter

Program:

$1	
N13	Sequential numbering
T1313	Select tool
M3 S1=1800	The main shaft is 1800
G0 G99 X0.0	Quickly locate to X0.0,
G0 Z-1.0	Quick positioning to Z-1.0
G83 Z8.0 Q2.0 F0.05	Drilling cycle, depth 8.0, each cut depth 2.0
G80	Cancel the drilling cycle
G0 Z-2.0	Retracting tool
G0T0	Cancel cutting
M01	Selective stop

8.5 Tapping processing

Explanation:

When tapping processing, wait for the spindle speed to arrive before the tapping process.

program:

$1	
N12	Sequential numbering
T1212	Select tool
M3 S1=400	Spindle 400 turns positive
G4 X0.4	Pause for 0.4 seconds and wait for spindle speed to reach
G0 G99 X0.0	Quickly locate to X0.0
G0 Z-1.0	Quick positioning to Z-1.0
G84 Z6.5 Q1.5 F0.8	Tapping cycle, depth 6.5, each cut depth 1.5
G80	Cancel the tapping cycle
G0 Z-2.0	Retracting tool
G0T0	Cancel cutting
M01	Selective stop

8.6 Outside diameter of coarse vehicle

Explanation:

The feed value can be increased appropriately during rough running.

program:

$1	
N2	Sequential numbering
T0202	Select tool
M3 S1=2000	Main axis 2000 turns positive
G0 G99 X30.0	Quickly locate to X30.0,
G0 Z-1.0	Quick positioning to Z-1.0
G1 X14.1 F0.5	Cut at F0.5 to X14.1
G1 Z14.4 F0.15	Cut at F0.15 to Z14.4
G1 X16.0	Cut to X16.0

G1 X17.1 W0.55 F0.05	Working chamfer
G0 Z-0.1	Back knife to Z-0.1
G1 X-0.5 F0.08	Cut at F0.08 to X-0.5
G1 X8.2	Cut to X8.2
G1 X10.1 W0.95 F0.05	Working chamfer
G1 Z4.9 F0.15	Cut at F0.05 to Z4.9
G1 X12.8	Cut to X12.8
G1 X14.1 W0.65 F0.05	Working chamfer
G0 Z-2.0	Back knife to Z-2.0
G0 X20.0	Back knife to X20.0
G0T0	Cancel cutting
M01	Selective stop

8.7 Lathe tooth processing

description:

program:

$1	
N03	Sequential numbering
T0303	Select tool
M3 S1=1000	Main axis 1000 turns positive
M120	Wait for the spindle speed to arrive
G0 G99 X19.0	Quickly locate to X19.0,
G0 Z3.0	Quickly locate to Z3.0
G1 X14.5 F2.0	X-axis positioning position
G92 X13.7 Z11.5 F1.0	Starting tooth
X13.5	Car tooth
X13.4	Car tooth
X13.3	Car tooth
X13.2	Car tooth
X13.1	Car tooth

X13.0	Car tooth
X12.9	Car tooth
X12.85	Car tooth
X12.85	Car tooth
G0 X20.0	X-axis back tool
G0 Z-2.0	Z-axis retracting tool
G0T0	Cancel cutting
M01	Selective stop

8.8 Outside diameter of finishing car

description:

program:

$1	
N04	Sequential numbering
T0404	Select tool
M3 S1=2000	Main axis 2000 turns positive
G0 G99 X19.0	Quickly locate to X19.0
G0 Z0.0	Quickly locate to Z0.0
G1 X14.1 F0.5	Cut at F0.5 to X14.1
G1 X-0.5 F0.065	Cut to X-0.5 at F0.065
G1 X8.1	Cut to X8.1
G1 X10.0 W0.95 F0.05	Working chamfer
G1 Z5.0 F0.07	Cut to Z5.0 at F0.07
G1 X12.7	Cut to X12.7
G1 X14.0 W0.65 F0.05	Working chamfer
G1 Z14.5 F0.07	Cut at F0.07 to Z14.5
G1 X12.7	Cut to X12.7
G1 X14.1 W0.7 F0.05	Working chamfer
G0 X20.0	Retracting tool
G0T0	Cancel cutting

M01	Selective stop

8.9 Groove processing

description:

program:

$1	
N5	Sequential numbering
T0505	Select tool
M3 S1=2000	Main axis 2000 turns positive
G0 G99 X19.0	Quickly locate to X0.0,
G0 Z12.0	Quickly locate to Z12.0
G1 X14.1 F0.5	Cut at F0.5 to X14.1
G1 X12.1 F0.065	Cut F0.065 to X12.1
G0 X14.5	Quickly locate to X14.5
G0 Z11.45	Quickly locate to X11.45
G1 X14.1 F0.15	Cut at F0.15 to X14.1
G1 U-1.1 Z12.0 F0.05	Cut at F0.05 to X13.0, Z12.0
G1 X12.0	Cut to X12.0
G0 X14.5	Quickly locate to X14.5
G0 Z12.4	Quickly locate to Z12.4
G1 X14.1 F0.15	Cut feed at F0.15 to X14.1
G3 U-0.8 Z12.0 R0.4 F0.05	Fillet finishing
G1 X12.0	Cut to X12.0
G0 X20.0	Retracting tool
M05	Spindle stop
G0T0	Cancel cutting
M01	Selective stop

8.10 Milling

description:

program:

$1	
N9	Sequential numbering
M05	Spindle stop
M80	C-axis function is enabled
M71	Specify side milling
T0909	Select tool
M33 S3=2000 G98	Side power cutter 2000 turn forward
G0 C0.0	Spindle C axis is positioned to 0.0°
G0 X19.0 Y14.0	Quickly locate X19.0, Y14.0
G0 Z19.0	Quickly locate to Z19.0
G1 G98 X14.85 F1000	Cut feed to X14.85
G1 Y-14.0 F150	Cut feed to Y-14.0
G0 C60.0	Axis C is positioned to 60.0°
G1 Y14.0 F150	Cut feed to Y14.0
G0 C120.0	Axis C is positioned to 120.0°
G1 Y-14.0 F150	Cut feed to Y-14.0
G0 C180.0	Spindle C axis is positioned to 180.0°
G1 Y14.0 F150	Cut feed to Y14.0
G0 C240.0	Axis C is positioned to 240.0°
G1 Y-14.0 F150	Cut feed to Y-14.0
G0 C300.0	Axis C is positioned to 300.0°
G1 Y14.0 F150	Cut feed to Y14.0
G0 X20.0	Retracting tool
M81	C-axis is enabled and closed
M70	Specified spindle machining
M35	Stop the side power cutter
G0T0	Cancel cutting
M01	Selective stop

8.11 Machining side center hole

description:

program:

$1	
N8	Sequential numbering
M05	Spindle stop
M80	C-axis function is enabled
M71	Specify side milling
T0808	Select tool
M33 S3=2000 G98	Main axis 2000 turns positive
G0 C0.0	Spindle C axis is positioned to 0.0°
G0 X19.0 Y0.0	Quickly locate to X19.0, Y0.0
G0 Z16.0	Quickly locate to Z16.0
G1 G98 X15.1 F1000	X axis is positioned at position X15.1
G87 X11.0 Q1.5 F150	Lateral drilling is processed to X11.0, each cutting depth is 1.5
G80	Cancel hole circulation
G0 X19.0	Quickly locate to X19.0
G0 C60.0	Axis C is positioned to 60.0°
G1 G98 X15.1 F1000	X axis is positioned at position X15.1
G87 X11.0 Q1.5 F150	Lateral drilling is processed to X11.0, each cutting depth is 1.5
G80	Cancel hole circulation
G0 X19.0	Quickly locate to X19.0
G0 C120.0	Axis C is positioned to 120.0°
G1 G98 X15.1 F1000	X axis is positioned at position X15.1
G87 X11.0 Q1.5 F150	Lateral drilling is processed to X11.0, each cutting depth is 1.5
G80	Cancel hole circulation
G0 X19.0	Quickly locate to X19.0
G0 C180.0	Spindle C axis is positioned to 180.0°

G1 G98 X15.1 F1000	X axis is positioned at position X15.1
G87 X11.0 Q1.5 F150	Lateral drilling is processed to X11.0, each cutting depth is 1.5
G80	Cancel hole circulation
G0 X19.0	Quickly locate to X19.0
G0 C240.0	Axis C is positioned to 240.0°
G1 G98 X15.1 F1000	X axis is positioned at position X15.1
G87 X11.0 Q1.5 F150	Lateral drilling is processed to X11.0, each cutting depth is 1.5
G80	Cancel hole circulation
G0 X19.0	Quickly locate to X19.0
G0 C300.0	Axis C is positioned to 300.0°
G1 G98 X15.1 F1000	X axis is positioned at position X15.1
G87 X11.0 Q1.5 F150	Lateral drilling is processed to X11.0, each cutting depth is 1.5
G80	Cancel hole circulation
G0 X19.0	Quickly locate to X19.0
G0X30.0	Back knife to safe position
M81	C-axis is enabled and closed
M70	Specify the spindle as the machining mode
M35	Power head stop
G0T0	Cancel cutting
M01	Selective stop

8.12 Side drilling

Explanation:

Procedure:

$1	
N17	Sequential numbering
M05	Spindle stop
M80	C-axis function is enabled

M71	Specify side milling
T0707	Select tool
M33 S3=2000	Main axis 2000 turns positive
G0 C0.0	Spindle C axis is positioned to 0.0°
G0 G98 X19.0	Quickly locate to X19.0, Y0.0
Z16.0	Quickly locate to Z16.0
G1 G98 X15.1 F1000	Cut feed to X15.1
G87 X5.0 Q1.5 F150	The side drilling is processed to X5. Each processing depth is 1.5
G80	Cancel hole circulation
G0 X19.0	Quickly locate to X19.0
G0 C60.0	Axis C is positioned to 60.0°
G1 G98 X15.1 F1000	Cut feed to X15.1
G87 X5.0 Q1.5 F150	The side drilling is processed to X5. Each processing depth is 1.5
G80	Cancel hole circulation
G0 X19.0	Quickly locate to X19.0
G0 C120.0	Axis C is positioned to 120.0°
G1 G98 X15.1 F1000	Cut feed to X15.1
G87 X5.0 Q1.5 F150	The side drilling is processed to X5. Each processing depth is 1.5
G80	Cancel hole circulation
G0 X19.0	Quickly locate to X19.0
G0 C180.0	Spindle C axis is positioned to 180.0°
G1 G98 X15.1 F1000	Cut feed to X15.1
G87 X5.0 Q1.5 F150	The side drilling is processed to X5. Each processing depth is 1.5
G80	Cancel hole circulation
G0 X19.0	Quickly locate to X19.0
G0 C240.0	Axis C is positioned to 240.0°
G1 G98 X15.1 F1000	Cut feed to X15.1
G87 X5.0 Q1.5 F150	The side drilling is processed to X5. Each processing depth is 1.5

G80	Cancel hole circulation
G0 X19.0	Quickly locate to X19.0
G0 C300.0	Axis C is positioned to 300.0°
G1 G98 X15.1 F1000	Cut feed to X15.1
G87 X5.0 Q1.5 F150	The side drilling is processed to X5. Each processing depth is 1.5
G80	Cancel hole circulation
G0X30.0	Back knife to safe position
M81	C-axis is enabled and closed
M70	Specify the spindle as the machining mode
M35	Power head stop
G0T0	Cancel cutting
M01	Selective stop

8.13 Machining of rear outer diameter

Explanation:

Procedure:

$1	
N6	Sequential numbering
T0606	Select tool
M3 S1=2000	Main axis 2000 turns positive
G0 G99 X19.0	Quickly locate to X19.0
G0 Z17.6	Quickly locate to Z17.6 position
G1 X17.1 F0.5	Cut at F0.5 to X17.1 position
G1 X14.1 F0.065	Cut at F0.65 to X14.1
G1 X13.0 Z23.78 F0.05	Cut at F0.05 to X13.0, Z23.78
G1 U-1.54 W0.77 F0.025	Cut to U-1.54, W0.77 in F0.025 increments
W0.2	Incremental cut to W0.2
G0 X20.0	Retracting tool

G0T0	Cancel cutting
M01	Selective stop

8.14 Cut off and connect the auxiliary shaft

Explanation:

After cutting off, the auxiliary shaft can be connected to complete the process of the auxiliary shaft, and the material is discharged to the receiving box or conveyor belt.

For the consideration of processing efficiency, the processing procedure and discharge of the auxiliary shaft part are written before receiving the material, and the auxiliary shaft can be processed at the same time when the main shaft is processed. In the actual processing of different products, please reasonably allocate the main shaft, auxiliary shaft part of the processing process, in order to maximize the processing efficiency.

Program:

$1		**$2**	
T0101	Select tool	T2020	The principal and auxiliary axes are concentric
M150	The chuck loosens the spindle to allow rotation	G0X0.0	X axis fast positioning
M3 S1=2000	Positive spindle rotation	G0Z0.0	Z-axis fast positioning
M104 S2=2000	Counteraxis inversion	;	
G0 G99 X19.0	Fast positioning	;	
G0 Z26.6	Fast positioning	;	
G114.1	Enable the main and auxiliary axes at the same time	;	
G04.1P10	Wait synchronization	G04.1P10	Wait synchronization
;		M150	Allow the loose clip to rotate
;		M111	The countershaft chuck is released
;		G0 G98 Z-140.0	Fast positioning
;		G1 Z-158.0 F2000	Advance to the receiving position
;		G4 X0.3	wait

$1		$2	
;		M110	Auxiliary shaft chuck clamping
;		G4 X0.2	wait
G04.1P11	Wait synchronization	G04.1P11	Wait synchronization
G1 X14.1 F0.15	Cut off	;	
G1 X-2.0 F0.065	Cut off	;	
G113	Close main and auxiliary axes at the same time	;	
G04.1P12	Wait synchronization	G04.1P12	Wait synchronization
M150	Allow the loose clip to rotate	G28 W0.0	Z2 Return to the safe position
M51	Test refuelling	M105	Countershaft stop
M151	Disable loose clip rotation		
M5	Spindle stop	;	
M9	Cutting oil shutdown	;	
M11	The spindle chuck is loosened	;	
M97	Tool life count opens	;	
G04.1P13	Wait synchronization	G04.1P13	Wait synchronization
M99	End of program	M99	End of program

8.15 Machining the outer diameter of the back

Explanation:

Procedure:

$2	
N21	Sequential numbering
T2121	Select tool
M104 S2=2000	Reverse the secondary spindle at 2000 RPM
G0 G99 X19.0	Quickly locate to X19.0
G0 Z-7.5	Quick positioning to Z-7.5

G1 X17.1 F0.5	Cut to X17.1
G1 U-1.0 Z-7.0 F0.065	Cut to X16.6Z-7.0
G1 X14.0 F0.05	Cut to X14.0
G1 X12.89 Z-0.77 F0.025	Cut to x 12.89z-0.77
G1 U-1.54 Z0.0	Cut to X12.12Z0.0
G1 X-0.5 F0.06	Cut to X-0.5
G0 Z2.0	Retracting tool
G28 W0.0	Return to zero Z
G0T0	Cancel cutting
M01	Selective stop

8.16 Drilling holes on the rear side

Explanation:

Procedure:

$2	
N22	Sequential numbering
T2222	Select tool
M103 S2=1800	Subspindle 1800 turns positive
G0 G99 X0.0	Quickly locate to X0.0
G0 Z-1.0	Quick positioning to Z-1.0
G83 Z-5.0 Q1.0 F0.05	The drilling is processed to Z-5.0 with a depth of 1.0 each time
G80	Cancel hole circulation
G0 Z2.0	Retracting tool
G28 W0.0	Return to zero Z
G0T0	Cancel cutting
M01	Selective stop

8.17 Machining the inner hole on the back

Explanation:

Procedure:

$2	
N23	Sequential numbering
T2323	Select tool
M103 S2=2000	Subspindle 2000 turns positive
G50 U-5.0	Tool width
G0 G99 X5.9	Quickly locate to X5.9
G0 Z1.0	Quickly locate to Z1.0
G1 Z0.1 F0.15	Cut to Z0.1
G1 X5.4 W-0.25 F0.035	Cut to X5.4, Z-0.15
G1 Z-5.0 F0.05	Cut to Z-5.0
G1 U-0.5 F0.025	Cut to X5.15
G1 W0.2	Cut to Z-4.8
G0 Z2.0	Retracting tool
G28 W0.0	Return to zero Z
G50 U5.0	Cancel tool width
G0T0	Cancel cutting
M01	Selective stop

8.18 Rear grooving

Explanation:

Procedure:

$2	
N24	Sequential numbering
T2424	Select tool
M103 S2=2000	Subspindle 2000 turns positive
G50 U-4.0	Tool width
G0 G99 X5.0	Quickly locate to X5.0
G0 Z1.0	Quickly locate to Z1.0
G1 Z-2.0 F0.5	Cut to Z-2.0
G1 X6.2 F0.065	Cut to X6.2
G0 X5.0	Quickly locate to X5.0
G0 Z-2.25	Quick positioning to Z-2.25

G1 X5.35 F0.1	Cut to X5.35
G1 U0.5 Z2.0 F0.03	Cut to X5.6Z 2.0
G1 X6.3 F0.05	Cut to X6.3
G0 X5.0	Quickly locate to X5.0
G0 Z-1.75	Quick positioning to Z-1.75
G1 X5.35 F0.1	Cut to G1 X5.35
G1 U0.5 Z2.0 F0.03	Cut to X5.6Z 2.0
G1 X6.3 F0.05	Cut to X6.3
G0 X5.0	Retracting tool
G0 Z2.0	Retracting tool
G28 W0.0	Return to zero Z
G50 U4.0	Cancel tool width
G0T0	Return to zero Z
M01	Return to zero Z

8.19 Discharge materials

Explanation:

For the convenience of programming, the position of the discharge port is defined as the tool No.25, which can be positioned to the discharge port.

Procedure:

$2	
N25	Sequential numbering
T2525	Select tool # 25 to locate the outlet
M105	The secondary spindle stops rotating
G0 X0.0	Quickly locate to X0.0
G0 Z-30.0	Quick positioning to Z-30.0
G4 X0.4	Pause 0.4 seconds
M58	The back shaft is pushed forward
M16	Back shaft blow open
G4 X0.3	Pause for 0.3 seconds
M59	Back shaft jacking back
M17	Back shaft blow off
G4 X0.2	Pause 0.2 seconds

G28 W0.0	Return to zero on the z-axis
M01	Selective stop

Chapter IX Maintenance

In order to keep the machine in good running condition for a long time, daily and regular maintenance should be carried out regardless of the mechanical failure.

9.1 Routine maintenance

Routine maintenance items are as follows:

Maintenance item	Maintenance content	Maintenance interval
Cutting tank	Level check and cutting oil refill. Chip cleaning	daily
Cutting oil plant	Check cutting oil switch and line. Cutting oil filter cleaning	daily
Lubricating oil installation	Lubricating oil level check and recharge	daily
Operating panel	Alarm message confirmation, keep operation panel clean	daily
Pneumatic device	Air pressure inspection and adjustment, drainage device inspection	daily

9.2 Routine maintenance

Regular maintenance items are as follows:

Maintenance item	Maintenance content	Maintenance interval
Spindle chuck	Clamping force check, parts cleaning	monthly
Countershaft chuck	Clamping force check, parts cleaning	monthly
Rotating guide bush	Parts cleaning	monthly
Bearing in all parts	Calorific value check	monthly
Pneumatic device	Starting line check, pressure gauge check	monthly
Cooling fan and strainer	Air volume check, filter cleaning	monthly
Safety door switch	Status confirmation	Every 6

		months
Emergency stop device	Function status verification	Every 6 months
Sliding parts of main and auxiliary axes	Abrasion inspection	Every 6 months
Synchronous belt	Check for tension and wear	Every 6 months
All lines, pipelines	Check whether it is properly fixed and damaged	Every 6 months
Connecting wire sponge	Check the protection	Every 6 months
Leakage switch	Confirmation of leakage action	Every 6 months

9.3 List of machine consumables

Due to mechanical reasons, sometimes it is necessary to use some parts with shorter life than other general parts. These parts are used as consumable parts for necessary regular spot inspection. If bad conditions occur, they must be replaced in time.

Name of consuming part	Spot detection period	Replacement standard
Spindle chuck	Use it continuously for 3 months	Wear, hole damage, abnormal loss
Counter shaft chuck	Use it continuously for 3 months	Wear, hole damage, abnormal loss
Spindle gripper	Use it continuously for 3 months	Abnormal wear
Counter shaft pick claw	Use it continuously for 3 months	Abnormal wear
Synchronous wheel belt	Use it continuously for 6 months	Damage, abnormal wear

Chapter X Troubleshooting

10.1 Contents and handing of system alarm

Attached Table I MLC alarm cause and disposal

The following is an analysis of the alarm causes of MLC alarm No. 65-114 and suggestions for solutions, please refer to

R44.0 alarm	**Spindle part alarm**
description	Spindle holding brake or spindle not clamping prohibit spindle operation.
Possible cause	The spindle motor is not enabled, and the spindle clamping cylinder is not ventilated.
Elimination method	1. The motor is not enabled before the spindle starts running. 2. Check the air source.3. Check the cylinder sensor.
R44.1 Alarm	**Dorsal axis section alarm**
description	The back shaft is in brake or the jacking has been jacked out to prohibit the spindle from running.
Possible cause	The back shaft motor is enabled and the ejecting cylinder has been ejecting.
Elimination method	1. Check whether the motor is enabled or not.2. Check the ejector cylinder sensor. 3. Check whether the electromagnetic valve of ejector cylinder is normal. Check air source.
R44.2 Alarm	**Spindle part alarm**
description	Spindle opening anomaly.
Possible cause	When the main shaft is in operation, the cylinder of the main shaft is released.
Elimination method	1. Check whether the M code, M10.M11, is incorrectly entered. 2. Check the air source. 3. Check whether the release cylinder and solenoid valve are faulty.

R44.3 Alarm	Spindle part alarm
description	The spindle is abnormally clamped
Possible cause	Spindle clamping cylinder stuck, cylinder sensor failure
Elimination method	1. Check whether the input of M code,M11, M10 is wrong; 2. Check whether the solenoid valve of the cylinder inductor is faulty
R44.4 Alarm	**Dorsal axis section alarm**
description	Abnormal dorsal axis release
Possible cause	Back shaft in operation spindle release cylinder action
Elimination method	1. Check whether the input of M code,M110,M111 is wrong; 2. Check the air source; 3. Check whether the release cylinder and solenoid valve are faulty
R44.5 Alarm	**Dorsal axis section alarm**
description	Abnormal back axle clamping
Possible cause	Back shaft clamping cylinder stuck, cylinder sensor failure
Elimination method	1. Check the M code. Whether M111, M110 input error,2. Check the cylinder, cylinder inductor solenoid valve failure
R44.6 Alarm	**Spindle part alarm**
description	Spindle speed anomaly
Possible cause	The check range of spindle speed is set to a small value

Elimination method	1. Check the value set for parameter 3415. If the value is small, set it to a larger value
R44.7 Alarm	**Dorsal axis section alarm**
description	The speed of the back shaft is abnormal
Possible cause	The check range of spindle speed is set to a small value
Elimination method	1. Check the set value of parameter 3415; 2. Check whether the synchronization instruction is normal in the program
R44.8 Alarm	**Machine tool part**
description	Emergency stop
Possible cause	The emergency stop switch is pressed. The line is abnormal
Elimination method	1. Check whether the emergency stop switch is pressed down, check whether the line is broken, corresponding to I. O board I point I (2)
R44.9 Alarm	**Spindle part**
description	Do not rotate C1 in the spindle holding brake
Possible cause	In enabling the spindle motor, there is the action of rotating the C1 spindle
Elimination method	1. The main shaft electromagnetic brake is held in the brake, and the electromagnetic brake can be loosened to rotate the main shaft
R44.10 Alarm	**Dorsal axial part**
description	Do not rotate C2 in the back shaft holding brake
Possible cause	In enabling the back shaft motor, there is the action of rotating C1 main shaft
Elimination method	1. The back shaft electromagnetic brake holds the brake, and the electromagnetic brake can be loosened to rotate the back shaft
R44.11 Alarm	**Machine tool part**
description	The main circuit is not connected
Possible cause	Contactor failure

Elimination method	1. Check whether the contactor works properly. 2. Check whether input I (3) to the controller I.O board is normal
R44.12 Alarm	**Spindle part**
description	It is forbidden to run the spindle without clamping the spindle chuck
Possible cause	Spindle chuck cylinder stuck, cylinder sensor failure
Elimination method	1. Check whether the cylinder is stuck; 2. Check whether the cylinder sensor is normal. 3. Whether the cylinder solenoid valve works normally
R44.13 Alarm	**Spindle part**
description	Spindle chuck is forbidden to operate at high speed
Possible cause	Misoperation, insufficient air pressure
Elimination method	1. Check whether the air pressure is normal and whether the program instruction is wrong. 2. Whether the cylinder works normally
R44.14 Alarm	**Dorsal axial part**
description	It is forbidden to run the back shaft without clamping the back shaft chuck
Possible cause	Spindle chuck cylinder stuck, cylinder sensor failure
Elimination method	1. Check whether the cylinder is dead. 2. Whether the cylinder sensor is normal. 3. Whether the cylinder solenoid valve works normally
R44.15 Alarm	**Dorsal axial part**
description	The back axle chuck is forbidden to operate at high speed
Possible cause	Misoperation, insufficient air pressure

Elimination method	1. Check whether the air pressure is normal and whether the program instruction is wrong. 2. Whether the cylinder works normally
R45.6 Alarm	**Receiving part**
description	Receiving failed to push out
Possible cause	Receiving cylinder stuck, cylinder inductor failure
Elimination method	1. Check whether the cylinder is locked. 2. Whether the cylinder sensor is normal. 3. Whether the cylinder solenoid valve works normally
R45.7 Alarm	**Receiving part**
description	Receiving failed to return
Possible cause	Receiving cylinder return stuck, cylinder return inductor failure
Elimination method	1. Check whether the cylinder is stuck. 2. Whether the cylinder sensor is normal. 3. Whether the cylinder solenoid valve works normally
R45.8 Alarm	**Dorsal axial part**
description	Back shaft ejector abnormal
Possible cause	Ejector cylinder stuck, cylinder sensor failure
Elimination method	1. Check whether the cylinder is stuck. 2. Whether the cylinder sensor is normal. 3. Whether the cylinder solenoid valve works normally
R45, 9 alarm	**Dorsal axial part**
description	Back shaft jacking back abnormal
Possible cause	Ejector return cylinder stuck, cylinder sensor failure

Elimination method	1. Check whether the cylinder is stuck. 2. Whether the cylinder sensor is normal. 3. Whether the cylinder solenoid valve works normally
R45.10 Alarm	**Dorsal axial part**
description	It is forbidden to push material in the back shaft clamping
Possible cause	The back shaft cylinder is in the clamping state
Elimination method	1. Check whether the clamping cylinder is normal. 2. Check whether the pushing cylinder is normal. 3. Check whether the cylinder inductor and solenoid valve are normal
R45.11 Alarm	**Dorsal axial part**
description	It is forbidden to tighten the back shaft in the push material
Possible cause	Cylinder solenoid valve failure, operation error
Elimination method	1. Check whether the pushing cylinder is normal. 2. Check whether the clamping cylinder is normal. 3. Check whether the cylinder inductor and solenoid valve are normal
R45.12 Alarm	**Auxiliary part**
description	The feeder is abnormal
Possible cause	The feeder is faulty and the line is abnormal
Elimination method	1. Check whether the feeder works normally. 2. Check whether the line between the machine tool and the feeder is normal
R45.13 Alarm	**Auxiliary part**
description	It is forbidden to start CNC if lubricating oil is seriously insufficient
Possible cause	The lubricating oil pump fails, the oil is insufficient, and the line is abnormal

Elimination method	1. Check lubricating oil. 2. Check whether the lubricating oil pump is faulty. 3. Check whether the oil pump line is broken
R45.14 Alarm	**Auxiliary part**
description	The feeder refueling time out, not according to the specified time to complete refueling
Possible cause	The feeder does not complete refueling within the specified time, and the feeder fails
Elimination method	1. Check whether parameter 3413 is incorrectly set. 2. Check whether the feeder is faulty. 3. Check whether the cable is normal
R46.00 Alarm	**Auxiliary part**
description	Electric cabinet refrigeration alarm (model selection)
Possible cause	The refrigerator in the electric cabinet is faulty. Line anomaly
Elimination method	1. Check whether the refrigerator in the electric cabinet has a fault alarm. 2. Check whether the line between the refrigerator and the machine tool is abnormal
R46.2 Alarm	**Dorsal axial part**
description	Z2 shaft pull material overload
Possible cause	Spindle clamp is not loosened,Z2 shaft motor failure
Elimination method	1. Check whether the program has errors. 2. Check whether the spindle clip is loosened. 3. Check the Z2 shaft motor. Whether the driver is faulty
R46.3 Alarm	**System part**
description	Line overload detection alarm
Possible cause	Trip, instantaneous current is too large

Elimination method	1. Check whether the circuit breaker trips. 2. Check whether the instantaneous current of the heavily loaded device is too large. 3. The circuit breaker is faulty
R46.4 Alarm	**System part**
description	Fan temperature detection alarm
Possible cause	The fan is overloaded, the temperature is abnormal, and the fan is damaged
Elimination method	1. Check whether the fan is stuck. 2. Check whether the temperature sensor is damaged. 3. Replace the fan module
R46.5 Alarm	**System part**
description	Cutting level deficiency alarm
Possible cause	Serious shortage of cutting fluid, abnormal cutting fluid inductor
Elimination method	1. Check whether there is cutting fluid. 2. Check whether the cutting fluid sensor is normal. 3. Replace the cutting fluid sensor
R46.6 Alarm	**System part**
description	High oil temperature alarm for oil cooling
Possible cause	The oil cooler is faulty
Elimination method	1. Replace the oil cooler
R46.7 Alarm	**System part**
description	Disconnect the detection exception alarm
Possible cause	The material is not cut off, and the sensor is stuck

Elimination method	1. Check whether the material is cut off. 2. Check whether the cut-off sensor is stuck. 3. Replace the cut-off sensor
R46.8 Alarm	**System part**
description	Oil cooler low level alarm
Possible cause	Oil cooler too little oil, oil cooler failure
Elimination method	1. Check whether there is oil leakage in the tubing joint. 2. Check whether the oil cooler is faulty. 3. Replace the oil cooler
R46.9 Alarm	**Dorsal axial part**
description	Back shaft ejector stuck alarm
Possible cause	Back shaft jacking cylinder stuck, cylinder sensor is abnormal
Elimination method	1. Check whether any foreign matter is stuck in the cylinder. 2. Check whether the spindle chuck is abnormal. 3. Whether the cylinder sensor is abnormal
R46.10 Alarm	**Spindle part**
description	Spindle motor temperature is too high alarm waiting for cooling
Possible cause	The oil cooling mechanism is cold abnormally, and the temperature sensor of the spindle motor is abnormal
Elimination method	1. Check whether the oil cooling mechanism works properly. 2. Whether the temperature sensor of the spindle motor is abnormal
R46.11 Alarm	**Dorsal axial part**
description	Back shaft motor temperature is too high alarm waiting for cooling
Possible cause	The oil cooling mechanism is cold abnormally, and the temperature sensor of the spindle motor is abnormal

Elimination method	1. Check whether the oil cooling mechanism works properly. 2. Whether the temperature sensor of the back shaft motor is abnormal
R46.12 Alarm	**Auxiliary part**
description	Pump overload alarm
Possible cause	The heat protector of the water pump trips and the water pump is stuck
Elimination method	1. Check whether the heat protector of the water pump trips. 2. Check whether the current of the pump motor is too high because foreign objects are stuck in the water pump
R46.13 Alarm	**Auxiliary part**
description	Feed overload alarm
Possible cause	The heat protector of the feeding machine trips, and the feeding machine is stuck
Elimination method	1. Check whether the heat protector of the feeding machine trips. 2. Check whether there is any foreign matter stuck in the conveyor belt, resulting in excessive current of the motor
R46.15 Alarm	**Auxiliary part**
description	Abnormal air pressure alarm
Possible cause	The air pressure is too low. The pressure gauge is faulty
Elimination method	1. Check the air pressure. 2. Check whether the pressure gauge works properly. 3. Check whether the pressure gauge is faulty due to water inlet
R590.6 alarm	**Safety door section**
description	Safety door open prohibit start processing
Possible cause	Safety door is open, safety lock is faulty

Elimination method	1. Check whether the safety door is open. 2. Check whether the door lock is abnormal. 3. Check whether the circuit of the security door lock is broken
R590.7 alarm	**Safety door section**
description	Safety door open processing pause
Possible cause	Safety door is open, safety lock is faulty
Elimination method	1. Check whether the safety door is open. 2. Check whether the door lock is abnormal. 3. Check whether the circuit of the security door lock is broken
R590.8 alarm	**Auxiliary part**
description	The feeder is not automatically prohibited from starting
Possible cause	The feeder is not automatic
Elimination method	1. If the feeder is not automatic, check whether the I. O line is abnormal
R590.12 Alarm	**Interference region part**
description	Protection disposal in Y1 and Z2 interference
Possible cause	The coordinate of the interference point of axis Y1 is less than the coordinate of Z2 when it enters the interference region is less than the trigger limit of parameter 3433 (R812).
Elimination method	Modify parameters to ensure safety (3433), (3432)
R590.13 Alarm	**Interference region part**
description	Z1 and guide sleeve interference protection disposal
Possible cause	Z1 axis forward limit with guide jacket mode (R814)
Elimination method	Modify parameters to ensure safety (3434)
R590.14 Alarm	**Interference region part**

description	X2 and Z2 interference protection disposal
Possible cause	X2 axis interference point coordinate, when the current X2 coordinate is less than this value and Z2 coordinate is less than parameter 3442, enter the interference region trigger limit (R821)
Elimination method	Modify parameters to ensure safety (3441), (3442)
R590.15 alarm	**Interference region part**
description	Y1 and X1 interference protection disposal
Possible cause	X1 axis interference point coordinates, when the current X1 coordinate is less than this value and the Y1 coordinate is less than parameter 3440, enter the interference region trigger limit (R819)
Elimination method	Modify parameters to ensure safety (3439), (3440)

Alarm ID	**OP-001**	**Alarm title**	**【Shaft plate setting error, I/O function can not start normally】**
description	This alarm will be issued when the controller does not detect a hardware interrupt signal (old shaft card: send a signal at one interval, new shaft card: send a signal at 0.5ms).		
Possible cause	1. The controller parameters are incorrectly configured. 2. The CPU BIOS Settings are incorrect. 3. The shaft card is faulty or the finger switch is incorrectly set. 4. The shaft card is not in contact with the ISA SLOT or the shaft card is compressed to the ISA SLOT. 5. Memory is too low (less than 1MB).		
Elimination method	1. Check whether Pr1 of the controller corresponds to the correct hardware specifications (refer to the parameter help of the application manual). 2. Check whether the BIOS IRQ11 is Legacy ISA. 3. Check the base address switch of the shaft card. 4. Check whether the interrupt signal ISR on the first piece of shaft card has short circuit (open circuit for other shaft cards). 5. Replace the shaft card to avoid bad contact between the gold finger of the shaft card and the ISA slot due to foreign matter or oil dirt.		
Alarm ID	**OP-002**	**Alarm title**	**【The system parameter file fails to be accessed. The system cannot work】**

description	The file name of the commonly used system parameter is called PARAM. DAT and stored in the C: \CNC\APP directory of the first CF card. The backup system parameter file name is called PARAM. LKN and stored in the C: Under the directory \CNC\LKN, when the system parameter files used normally and backed up are found damaged or cannot be found at boot time, the system will re-establish the two files according to the internal parameter values, but if the system fails to establish the two files, the system will issue this alarm to warn the user.
Possible cause	1. The first CF card is faulty.
Elimination method	1. Remove the first CF card from the controller, insert it into the CF card reader, connect it to the PC, repair the CF card on the PC, and copy the backup parameters to the C: \CNC\APP directory of the controller. (The backup parameter file name is PARAM.LKN). 2. Replace the first CF card. (Please note that the first CF card must be installed with DOS startup system and CNC core software.

Alarm ID	**OP-003**	**Alarm title**	**【Processing data file access failed, the system cannot operate】**
description		1. The processing data file is named REGISTRY. DAT and stored in the C: \CNC\USER directory. 2. The first backup processing data file name is REGISTRY. LKN and is stored in the C: \CNC\LKN directory. 3. The second backup processing data file named REGISTRY. MIR is stored in the C: \CNC\MIR directory. 4. If the system parameter files used and backed up are damaged or cannot be found during startup, the system will recreate these three files. 5. However, if the system fails to create the three files, the system will issue this alarm to warn the user.	
Possible cause		The second CF card is faulty.	
Elimination method		If the fault persists, back up user data to a new CF card and replace the second CF card.	
Alarm ID	**OP-004**	**Alarm title**	**【If processing data is lost, please recalibrate the processing data before processing.】**

description	When the controller starts up, it will reload the system login file before last shutdown into the memory. After loading, it is found that the status log of last shutdown is processing Issue this alert if both the normal login file and the first backup file are corrupted during or load. (The normal login file name is REGISTRY. DAT, and the first backup login file name in the D: \CNC\USER directory is REGISTRY. MIR, and D: \CNC\MIR exists The second backup login file name in the directory is REGISTRY. LKN, which is stored in the D: \CNC\LKN directory)
Possible cause	1. The controller shuts down or loses power in the 'processing' state. 2. Processing data login file is corrupted. 3. The second CF card is faulty.
Elimination method	If the fault persists, back up user data to a new CF card and replace the second CF card.
Alarm ID OP-005	**Alarm title 【I/O Transmission error】**
description	This alert is issued when PIO5's Watch Dog fails.
Possible cause	1. The mainboard is overheating. // Check diagnostic variable 39. 2. PIO5 is faulty. 3. The controller ground cable is dry wound. 4. Main board failure. 5. The Pr5 configuration is incorrect. 6. No external 24V voltage is provided to RIO.
Elimination method	1. Check whether the fans in the chassis work properly 2. Replace PIO5. 3. Ensure that the machine is properly grounded. 4. Replace the mainboard. 5. The hardware framework used by the root is configured with the correct Pr5 parameters. 6. Provide external 24V voltage to RIO.
Alarm ID OP-006	**Alarm title [Fram CRC error]**
description	This alarm is generated when the I/O card hardware is faulty.
Possible cause	1. IO card fault 2. Fram CRC error 3. Fram is not initialized
Elimination method	or Replacing an I/O card Please update to versions later than 10.112.95 or 10.114.29

Alarm ID	OP-007	Alarm title	【The memory space is insufficient (less than 1 MB)】
description		This alarm is issued when the system memory is less than 1 MB.	
Possible cause		When the user uses too many parts and too many pictures when designing the controller CE man-machine picture, the controller makes the system memory consume too much and less than 1M when executing.	
Elimination method		Controller man-machine design needs to simplify the use of too many components and effectively use variable setting and component use to avoid excessive memory use.	
Alarm ID	**OP-008**	**Alarm title**	**【The memory space is seriously insufficient, less than 100K】**
description		This alarm is generated when the system memory is less than 100k.	
Possible cause		When the user uses too many parts and too many pictures when designing the controller CE man-machine picture, the controller makes the system memory consume too much and less than 100k when executing.	
Elimination method		Controller man-machine design needs to simplify the use of too many components and effectively use variable Settings and components to avoid excessive memory use.	
Alarm ID	**OP-009**	**Alarm title**	**【The difference timeout times exceeded 100】**
description		The number of diagnostic variable 54 (interpolation timeout times) exceeds 100. Interpolation timeout: the shaft card does not normally send out the command calculated for the last interval time. The system will ignore this interpolation to avoid the command leakage caused by the unsent command quantity being overwritten.	
Possible cause		1. The shaft is broken. 2. System resource exhaustion. 3. The interpolation time (3203) is set too short.	
Elimination method		1. Replace the axle card hardware. 2. Replace the mainboard with a good one. 3. Set the interpolation time to be larger.	
Alarm ID	**OP-010**	**Alarm title**	**【 The axis group number parameter is incorrectly configured】**
description		When the CNC main system axis group does not reach the group axis set by parameter, the controller will issue this alarm.	

Possible cause	Parameter setting error Example: Suppose the machine has three axes XYZ, whose axis group is set as follows Pr701 = Pr702 = Pr703 = 5 (XYZ axes all belong to the first axis group + the third axis group) Assume that Pr731 = 3 (the axis group of CNC main system is the first group axis, the second group axis and the third group axis). Since the axis group of each axis does not cover the second group axis, That is to say, when the second axis group of the main system has no axial control, the controller will issue this alarm
Elimination method	Check whether Pr701 to Pr716 matches Pr731

Alarm ID	**OP-011**	**Alarm title**	**【The pause point offset is incorrect】**
description		When multi-axis group machining is used, if there is an axis belonging to complex number axis group in different work coordinate system, the offset of the axis on different coaxial groups is different when the user stops modifying the work coordinate system in a single section due to improper operation, this situation will cause the control system can not issue the correct moving command; When this unreasonable situation occurs, the system will issue this alarm to warn the user.	
Possible cause		When changing the workpiece coordinate system of the axial direction, the user does not modify all the axis groups belonging to the axial direction. When changing the coordinate system of the workpiece of all the axially assigned axis groups, the change of each assigned axis group is different.	
Elimination method		Please make sure that the coordinate system of the workpiece in the axis group belonging to all the axis groups has been changed (refer to the axis group belonging to Pr701~716). Please confirm that the modification amount of the axis in the workpiece coordinate system of different axis groups is the same.	

Alarm ID	**OP-012**	**Alarm title**	**【Parameter 3219 The CF card used for setting is incorrect. Please check the CF card or system Settings!】**
description		When the controller is under Dos CNC, parameter 3219 is set to 3 and no second CF card (user expansion) is inserted. In order to avoid the error of registry.dat data capture due to CF card failure, the controller sends out	
Possible cause		1. The second CF card is not inserted into the system, or the second CF card is faulty and cannot be read. 2. Two CF cards are inserted into the system, but the two CF cards have only one sector.	

Elimination method		1. Check whether the second CF card exists or fails to be read. 2. The first piece is re-established as the factory standard state, that is, the state with two divided magnetic fields. 3. Change the parameter 3219 to 0 and use all C:.	
Alarm ID	**OP-013**	**Alarm title**	**【This software version and model cannot drive this controller hardware, please contact the supplier! 】**
description		When the controller model (ex: EZ, 10A..) The installed software version (ex: 10.115.x.. etc.), the controller will issue this alarm. Before this alarm is excluded, if the user moves the machine, the controller will appear following error alarm.	
Possible cause		1. The controller is installed to a software version that the hardware does not support, such as EZ Milling machine installation 10.114. 2. The mainboard or CF card or I/O card of the controller has been replaced at the same time, causing inconsistent hardware information. The model is reduced to EZ, so the original software version cannot be supported.	
Elimination method		Install the software version supported by this model, such as EZ milling machine 10.112 version, if you want to use a higher version, you need to open option1 "EZ2/3/4 upgrade 10A Software function ". Ask the new generation for a password to resolve hardware information discrepancies and reset the controller serial number.	
Alarm ID	**OP-014**	**Alarm title**	**【WinCE function option is not enabled, please contact the supplier! 】**
description		When the CE version software of the controller is installed and the WinCE function option is not enabled, the shaft card will not be able to issue commands, resulting in a tracking error alarm after the controller starts processing. Therefore, the controller will issue this alarm.	
Possible cause		Option2 of the controller "Controller upgrade WinCE system" is not enabled.	
Elimination method		1. Install back to Dos version. 2. Request password to new generation, open Option2" Controller upgrade WinCE system ". 3. The Super controller is not subject to this condition.	
Alarm ID	**OP-015**	**Alarm title**	**【The output contact is overloaded. Power off to check whether the cable distribution is abnormal】**

description		This alarm will be issued when the controller detects that the O-point output of the PLC exceeds the load. The 16 O-points in a group will be issued when the load of one of the O-points in this group exceeds 350mA.	
Possible cause		1. The load exceeds 350mA. 2. The output contact is short-circuited. 3. Machine leakage. 4. Load equipment aging.	
Elimination method		1. Check the load power consumption. 2. Verify that the output is not short-circuited. 3. Machine grounding. 4. Load replacement test.	
Alarm ID	**OP-016**	**Alarm title**	**【Parameter setting of straightness compensation function is incorrect】**
description		Due to straightness compensation function, at present, up to five sets of moving axis and compensation axis Settings are provided, and each group must input the compensation position of five moving axis and the compensation amount of the corresponding compensation axis. Therefore, when the system detects that the parameter setting of straightness compensation function is wrong, the controller will issue this alarm.	
Possible cause		1. Multiple groups of moving axes correspond to the same compensation axis. 2. The moving axis is also the compensation axis. 3. Turn on the motion axis compensation for straightness to decrease the compensation position. 4. Five compensation positions are not entered for the moving axis with straightness compensation enabled.	
Elimination method		1. Please check Pr.1481~Pr.1490 to match multiple groups of moving axes to different compensation axes. 2. Please check Pr.1481~Pr.1490 to set the moving axis as the compensation axis. 3. Check Pr.7001 to Pr.7025 and fill them into the non-decreasing compensation position order. 4. Please check Pr.7001~Pr.7025 and fill it into five compensation positions.	
Alarm ID	**OP-017**	**Alarm title**	**【The port number or I-point parameter corresponding to the handwheel is incorrectly set】**
description		Pr2021 to 2027 is incorrectly set.	

Possible cause	1. Port numbers configured for PR2021-2023 are not supported. 2. Two consecutive I points set by Pr2024 to 2027 are not supported.
Elimination method	Please refer to the parameter manual to make correct Settings based on the model of the controller.

Alarm ID	**OP-018**	**Alarm title**	**【The port number of the axis or spindle is incorrectly set】**
description		Port number of the axis or spindle of Pr21~ or Pr1621~ is incorrectly set.	
Possible cause		1. The axial port number set by Pr21~ is not supported. 2. When the variable frequency spindle is used (Pr1791~=0), the spindle port number set by Pr1621~ is not supported. 3. The serial controller uses variable-frequency spindle (Pr1791~=0), and the port number of the variable-frequency spindle (Pr1621~) shall not conflict with other serial servo port numbers (1-16).	
Elimination method		1. Set the correct port number by referring to the parameter manual. 2. Set Pr1621~ to a number that does not conflict with other serial port numbers (1-16), and 19 is recommended.	

Alarm ID	**OP-020**	**Alarm title**	**【M-code parameters conflict with each other】**
description		1. This alarm is generated when duplicate M codes are set for the parameters related to the following M codes, resulting in a set conflict. A. Parameter 3600 login interrupt type program call function M code. B. Parameters 3601~3610 Login M code call macro. C. Parameter 3804 Work piece count M code number set 2. This alert is issued when parameter 3600 Login interrupt type program call function M code is set to 1, 2, 29, 30, 97, 99, 99, 197, 198. (Same as M2/M30/M98/M99/M198 conflict).	
Possible cause		With the above description	
Elimination method		Modify the above several parameters so that they no longer conflict after restart after clear alarm.	

Alarm ID	**OP-021**	**Alarm title**	**【When using the previous processing data file, please check the processing data setting value】**
description		The processing data of the controller is stored in registry.dat, and there are two copies of registry.mir and registry. This alert is raised and the processing file name is cleared when the system is using registry.lkn.	
Possible cause		The CF card is damaged, causing the registry.dat and registry.mir files to be damaged.	

Elimination method		1. Perform magnetic drive scan for the CF card. 2. Reset or check whether the processing configuration data is correct. 3. Shut down and restart.	
Alarm ID	**OP-022**	**Alarm title**	**【The processing data file is reconstructed. Please recalibrate the processing data before processing】**
description		dat and backup files registry.mir and registry.lkn do not exist in the controller, and files need to be re-created.	
Possible cause		1. Replace the CF card. 2. The CF card has been formatted.	
Elimination method		1. Reconfigure processing data. 2. Shut down and restart.	
Alarm ID	**OP-023**	**Alarm title**	**【The processing is interrupted. Please recalibrate the processing data before processing.】**
description		When the controller starts the processing, the flag in the processing will be saved back to Fram or registry.dat, and cleared when the processing is back to ready. This alarm is issued when a shutdown is restarted and the flag in process is not cleared.	
Possible cause		The processing is interrupted.	
Elimination method		1. Do magnetic drive scan of CF card. 2. Check whether the processing setting data is correct. 3. Shut down and restart..	
Alarm ID	**OP-024**	**Alarm title**	**【Failed to write the processing data file. Please recalibrate the processing data before processing】**
description		The controller issues this alert when it failed to write the registry.dat or registry.mir file for the last time before shutdown.	
Possible cause		1. The CF card is damaged. Procedure 2. The registry.dat or registry.mir file is set to read-only. 3. The resources of the file processing device in the system are insufficient. As a result, file writing fails.	
Elimination method		1. Do magnetic drive scan of CF card. 2. Check whether the processing setting data is correct. 3. Shut down and restart.	
Alarm ID	**OP-025**	**Alarm title**	**【Failed to write the processing data file for many times. Please recalibrate the processing data before processing】**

description	This alarm is generated when the controller fails to write files to the registry.dat or registry.mir file more than 100 times before shutdown.
Possible cause	1. The CF card may be seriously damaged. Handle it as soon as possible. 2. The registry.dat or registry.mir file is set to read-only. 3. The resources of the file processing device in the system are insufficient. As a result, file writing fails.
Elimination method	1. Do magnetic drive scan of CF card. 2. Check whether the processing setting data is correct. 3. Shut down and restart.

Alarm ID	**OP-026**	**Alarm title**	**【The processing data file fails to be written seriously. Please recalibrate the processing data and perform magnetic drive scanning before processing】**
description		This alarm is issued when the controller has failed to write files to registry.dat or registry.mir for 100 consecutive times before shutdown.	
Possible cause		1. The CF card may be seriously damaged. Handle it as soon as possible. 2. The registry.dat or registry.mir file is set to read-only. 3. The resources of the file processing device in the system are insufficient. As a result, file writing fails.	
Elimination method		1. Do magnetic drive scan of CF card. 2. Check whether the processing setting data is correct. 3. Shut down and restart.	

Alarm ID	**OP-027**	**Alarm title**	**【The CNC main system axis group to be executed is not selected】**
description		This alarm is issued when CycleStart is triggered without selecting any CNC main system axis group.	
Possible cause		Please check the value of R21: 1. The selected axis group does not exist or does not belong to the CNC main system axis group. 2. No CNC main system axis group is selected (only when R21.11 is ON and the value is BIT).	
Elimination method		1. Please check whether there is logic error when PLC triggers R21 selection, resulting in failure to select the correct axis group. 2. Please select any existing CNC main system axis group through R21.	

Alarm ID	OP-055	Alarm title	【M3 information communication is abnormal】
description		MECHATROLINK-III message communication is abnormal	
Possible cause		1. The communication between the controller and the driver is unstable, resulting in a long time for sending and receiving messages 2. The drive does not support communication via Message	
Elimination method		1. Check the connection of communication cables between the controller and the driver to eliminate the cause of unstable communication 2. Verify that all connected drives support communication via M3 Message	

2 Axial alarm – MOT

Alarm ID	Pair-001	Alarm title	【A,B encoder feedback signal error】
description		A/B encoder feedback error is detected by shaft card only when shaft plate type of parameter No. 9 is set to 0(EMP2).	
Possible cause		When the shaft card is EMP2, the signal of A and B encoder will be automatically detected inside the shaft card. If the signal is wrong or there is A phase signal but no phase B signal, this alarm will be issued.	
Elimination method		Check the servo line or replace the shaft card.	

Alarm ID	Pair-001	Alarm title	【Error counter overflow】
description	Only when the shaft plate type of parameter No. 9 was set to 0(EMP2),4(PMC4),6(SERVO6), the shaft card detected the encoder back overflow.		
Possible cause	When the shaft plate type is set to 0(EMP2),4(PMC4),6(SERVO6), the shaft card will automatically detect the signal of A and B encoder. If the signal is wrong or the signal input is too large, the counter overflow will be sent.		
Elimination method	Check the servo line or replace the shaft card.		

Alarm ID	Pair-005	Alarm title	【The DDA instruction exceeds the rating】
description	The controller sends a high number of instructions, and the software calculates that more than 2,047 pulses must be sent in a tween time.		
Possible cause	1. The DDA software time (tween time, parameter 3203) is set too long. 2. You're moving too fast 3. The servo resolution is set too high 4. The amount of backgap compensation or pitch compensation is too large. 5. Enable the feedforward compensation function.		
Elimination method	1. Reduce the setting of software tween time (CNC parameter 3203), but it is recommended not to be lower than 2000 2. Reduce the motion speed for testing. If it is solved, lower the upper limit of motion speed (CNC parameters 461-480). 3. Reduce the servo resolution setting test (drive and CNC parameters 61~80) 4. If the mechanism compensation constant is set (CNC parameter 1401~1420), cancel the mechanism compensation for testing, if 5. If yes, set the appropriate mechanism compensation time constant 6. If there is feedforward compensation (CNC parameters 581~600), cancel the feedforward compensation for testing; if there is, reduce or turn off the feedforward compensation function 7. Please contact the staff of the machinery factory		
Advanced statement	1. For the relevant parameters and calculation description of DDA rule, please refer to the document: DDA Alarm Description Document. 2. In order to achieve the multi-axis synchronous control, the new controller adopts the mathematical method of DDA(Digital Differential Analyzer) and the Cycle Time of DDA 3. Set by parameter 3203, each axial instruction is allowed to send a maximum of 2047 pulse signals within a DDA Cycle Time. Beyond this range, the controller will send an alarm message of DDA instruction exceeding. If G68.2/G68.3 equioblique plane machining function is used, because the actual motion axis may be different from the axis specified by the machining program, in the adjustment of motion speed (CNC parameters 461-480), the axial parameters to be modified may be different from the actual motion axis. 4. It is recommended to use a string controller instead because there are no DDA restrictions at all.		

Alarm ID	Pair-006	Alarm title	【The power status of the drive is incorrect 】
description	If the controller is sending motion commands, the drive Power status error is detected (Servo Off or Power Off). (This alert only applies if Yaskawa Drive masks A.95x warnings).		
Possible cause	1. The rear voltage of the driver is unstable. 2. When the emergency stop is pressed, the power supply behind the drive is cut off. 3. The drive is faulty.		
Elimination method	1. Check whether the power supply status of the drive is abnormal. 2. Check whether the power cable of the driver is loose or broken. 3. Replace the drive.		

Alarm ID	Pair-007	Alarm title	【 Drive outage 】
description	The power input to the drive is disconnected		
Possible cause	1. Power off 2. The main power supply is turned off		
Elimination method	After ensuring that the controller is completely powered off, confirm the power input and restart the controller.		

Alarm ID	Pair-007	Alarm title	【Lost position command】
description	After the controller stops the output of an axial instruction for 1 second, it will check whether the error between the feedback instruction quantity and the output instruction quantity is within the predetermined error range at any time. If it is not within this range, there will be a lost position alarm.		
Possible cause	1. The mechanism movement is obstructed. 2. Unexpected Servo ON/OFF of the servo driver. 3. The data sent by the CPU board to the shaft card is lost (the CPU board has problems, the shaft card has problems, the CPU board and the shaft card are not in good contact). 4. The command transmission line from the controller to the servo driver is in bad contact or disconnected. 5. The controller does not set the servo driver alarm check, resulting in the controller still sends motion instruction when the driver is abnormal 6. Local interference. 7. The string axis encountered this alarm when seeking the origin, which may be improper setting of Pr901~ or Pr561~.		

Elimination method	1. Do not turn off the machine after the alarm occurs. First check whether the values of data 8,9 and 10 in the diagnostic function converge to 0 2. Check whether the lubrication system of the mechanism is good and whether the track lubrication is normal 3. Open the axial cover to check whether the axial movement is blocked by foreign bodies 4. Rotate the screw by hand to check whether the mechanism is stuck. (Drive load) Check the signal power supply or connection lines for the drivers servo-on and servo-off If the values of data No. 8,9, and 10 of the diagnostic function continue to remain unchanged, please re-search the origin (without restarting the machine). After the origin search, observe parameters No. 24,25,26 and 40. Parameters 41,42, if any of the parameters 24,25,26 is not zero, then there is a problem with the feedback loop If any of 40,41,42 is not zero, there is an instruction missing from the controller to the motor line If 24,25,26 and 40,41,42 are all non-zero, the signal is more likely to be interfered, which is specifically manifested in 8, 9, and 10 in processing The value of the sign parameter increases gradually. The specific fault point may be caused by bad contact between CPU board and shaft card. Can be replaced by CPU board and shaft card try Lower bottom Pr901~ or enlarge Pr561, the recommended values are as follows l Pr561: Straight spool set at 100; Set the rotation axis at 500 l Pr901 1. General use: axial and rotating axis set 3 2. Bus: linear spool set 300; Let's say the rotation axis is 200
Advanced statement	Advanced note: Parameters 561 to 580 are the check range of the lost location command 8[X-axis tracking error] 24[X-axis absolute position feedback] 40[X-axis absolute position life] 9[X-axis tracking error] 25[Y-axis absolute position feedback] 41[Y-axis absolute position life] 10[Z-axis following error] 26[Z-axis absolute position feedback] 42[Z-axis absolute position life]

Alarm ID	**Pair-009**	**Alarm title**	**【Driver alarm】**
description	The drive sends an alarm signal.		
Possible cause	Drive alarm is mostly caused by external causes, such as: drive temperature is too high; Encoder wiring error; Internal parameter setting is incorrect. Incompatible with servomotor; Servo drive alarm such as drive failure.		
Elimination method	Follow the alarm troubleshooting steps in the driver manual.		

Alarm ID	Pair-010	Alarm title	【The communication of the axial position command is abnormal】
description	The core program is abnormal when communicating with the shaft card. The software checks that the internal queue value of the shaft control IC is not zero.		
Possible cause	1. One axis card, but the parameters are set to two axis cards, and a servo axis points to the second axis card 2. One of the two shaft cards is faulty 3. If more than two axis cards and more than two axis cards IRQ11 Jump are plugged in, the diagnosis function No. 23 is not 100 4. Incorrect setting of timing pulse source parameters of shaft stuck (Pr.11		
Elimination method	1. Check whether parameter 11 and parameter 13 are consistent with the hardware specifications. 2. Check the shaft Jump setting. 3. Replace the shaft card for testing.		
Advanced statement	The core software checks whether the QUEUE FLAG value is correct at each fill time, and if it reads an incorrect value after running through some filters, it raises an alert and adds one to diagnostic 68.		

Alarm ID	Pair-011	Alarm title	【Driver communication exception】
description	This alert is issued when the communication of the string driver is abnormal.		
Possible cause	1. The drive office number and controller parameters do not correspond correctly. 2. The communication cables between the controller and the driver fall off unsecurely. 3. Communication wire quality problem, or there is noise interference. 4. M3 communication wire is an old version with poor noise resistance.		
Elimination method	1. Check that the driver parameters (dial) correspond correctly to the controller parameters (Pr21~). 2. Check the connections between the controller and the driver. 3. Check that the machine is properly grounded and the terminal driver communication port is equipped with terminal resistance to reduce noise interference. 4. Check whether there is a yellow label sticker with the word V4 on the M2 communication line. If no, replace the cable.		

Alarm ID	Pair-012	Alarm title	【The drive failed to find the origin】

description	This alert is issued when the string driver fails to find the origin.
Possible cause	The origin search method (Pr961) is incorrectly set or the drive does not support origin search.
Elimination method	Check whether the origin method is set correctly or whether the drive supports origin finding.

Alarm ID	**Pair-013**	**Alarm title**	**【 After the axial tuning fails, restart the controller and then JOG, MPG, and seek the origin】**
description	When a series of axial adjustment fails and the machine is not restarted, the function of the axial inching, hand wheel movement or origin searching is triggered.		
Possible cause	Serial axial tuning failed		
Elimination method	restart		

Alarm ID	**Pair-016**	**Alarm title**	**【The absolute encoder has been reset. Please remove battery abnormalities and power off the driver and restart it】**
description	Absolute encoder anomaly detected, reset has been performed, please power off the drive and restart.		
Possible cause	1. Power on the absolute encoder for the first time and set the encoder. 2. The power supply of the encoder is completely exhausted, and the position data is cleared. 3. After the driver is powered on, the voltage of the encoder battery is too low.		
Elimination method	1. Verify that the battery and connection status of the absolute encoder is normal. 2. Verify that the battery voltage is stable and meets the voltage specification required by the absolute encoder. 3. After completing the above check, power off the controller and drive and restart.		

Alarm ID	**Pair-017**	**Alarm title**	**【Exceed the first forward software travel limit】**
description	The axial mechanical coordinates exceed the forward travel limit set by Pr2401~Pr2440		
Possible cause	Bed movement exceeds set value		

Elimination method	1. Press the Reset axis to move in the negative direction. 2. Go out of the software stroke protection range

Alarm ID	**Pair-018**	**Alarm title**	**【The first negative software travel limit is exceeded】**
description	The axial mechanical coordinates exceed the negative software travel limit set by Pr2401~Pr2440		
Possible cause	Bed movement exceeds set value		
Elimination method	1. Press Reset 2. Axial forward movement, out of the software stroke protection range		

Alarm ID	**Pair-019**	**Alarm title**	**【Tracking error exceedance】**
description	Because of the servo characteristics, the positioning of the servo motor cannot immediately respond to the command of the controller, and there will be lagging phenomenon. When the lagging amount is not within the allowed range, the controller will issue an alarm.		
Possible cause	1. The mechanism movement is not smooth. 2. The cable is in poor contact. 3. The acceleration and deceleration time set by the controller parameters is too short. 4. Servo on off Relay is interfered. 5. Internal loop gain of driver parameters is too small (version after 10.116.10 and 10.117.10, the Kp of controller and driver will automatically synchronize, so this factor is not needed). 6. Wrong setting of resolution of parameter encoder or electronic gear ratio. 7. The driver or motor is faulty. 8. The encoder is abnormal or the cable between the encoder and controller is abnormal. 9. Diagnostic screen 23 is not 100.		
Elimination method	1. Lubrication test of adding lubricating oil to mechanism. 2. Use a three-purpose meter to measure whether the wire connection is normal. 3. Increase parameter 401 to set acceleration and deceleration time. 4. The machine runs empty, open the case to observe whether the relay Servo on off has abnormal pulsation. 5. The inner loop gain is too small. In the case of Mitsubishi drive, check Pr37. 6. Contact the machine manufacturer for processing.		

Advanced specification	Take the maximum value of the G00 speed set by the parameter and the seeking speed of the origin, divide it by the Kp set by the parameter, and multiply it by twice, that is, the setting range of the controller. Reasonable following error: Ferr = speed instruction/loop gain setting Alert allowed value = {max[(first stage velocity of each axis seeking origin),(G00 velocity of each axis)]/Kp}*2 Example: Speed 1000mm/min, loop gain 30, accuracy 1um, Ferr = 1000*1000÷60÷30=555 32[X-axis reasonable following error] 33[Y-axis reasonable following error] 34[Z-axis reasonable following error]

Alarm ID	**Pair-020**	**Alarm title**	**【Cannot switch back to control mode while on the move】**
description	When the motor is moved beyond zero speed within an interpolation time (No3203), the controller of the check window (Parameter 901~) will sound an alarm when the motor is stopped or the watch mode (C31~) is canceled		
Possible cause	1. Cancel instant artificial mobile console. 2. The driver gain is set poorly, causing the motor to jitter when the monitoring mode is cancelled. 3. Pr901 Set too small (especially in tandem axis).		
Elimination method	1. Avoid human movement 2. Check the driver's position loop gain and speed loop gain Settings 3. Enlarge Pr901 Settings		

Alarm ID	**Pair-021**	**Alarm title**	**【You have to start all over again】**
description	When MOT-0020 and MOT-0022 appear, the controller will issue an alarm.		
Possible cause	MOT-0020 [Cannot switch back to control mode while moving] or MOT-0022 [inaccurate origin position] when triggered		
Elimination method	Please exclude causes related to MOT-0020 [Failure to switch back to control mode while moving] or MOT-0022 [inaccurate origin position]		

Alarm ID	**Pair-022**	**Alarm title**	**【The origin position is not accurate】**
description	After the initial origin search is completed, the controller will either clear the servo command (diagnostic variable 40~) and mechanical coordinate (diagnostic variable 72~) to zero or set an offset according to Pr881~. The controller will issue this alarm if the difference between the servo command and 0 or Pr881~ is		

	equivalent to 0.1 revolutions more than the number of motor turns when the origin is sought again.
Possible cause	1. Motor origin signal is abnormal. 2. Loose origin block, coupling or bearing fixing seat. 3. Origin grid function (Pr941~) is not enabled. 4. In a series environment, when the axis is the main axis at the same time, the Settings of Pr881~ and Pr1771~ are different, resulting in the change of the origin position after searching the origin.
Elimination method	1. Move the motor in the same direction and observe whether the index position counter increases or decreases normally 2. Check whether the components of the mechanism are fixed correctly 3. If this is the reason, please set Pr886 and Pr1771 to the same value assuming that the sixth axial axis is set as the first spindle

Alarm ID	**Pair-023**	**Alarm title**	**【Severe tracking error exceeds】**
description	Because of the servo characteristics, the positioning of the servo motor cannot immediately respond to the command of the controller, and there will be lagging phenomenon, when the lagging amount is greatly beyond the allowable range.		
Possible cause	1. The servo motor moves out of control due to external force 2. The loop gain in the driver parameter is too small 3. The acceleration and deceleration time set by the controller parameter is too short 4. The encoder is abnormal or the route from the encoder to the controller is abnormal		
Elimination method	1. Check the external motion mechanism of the bed table 2. Check drive parameter Settings 3. Check the acceleration and deceleration Settings of each shaft, parameter 401, 541~560 4. Maintain good connection between encoder and servo driver		
Advanced specification	Take the maximum G00 speed and homing speed set by the parameter, divide by Kp set by the parameter, and multiply by four times, that is, the setting range of the controller. Reasonable following error: Ferr = speed instruction/loop gain setting Alert allowed value = {max[(first stage velocity of each axis seeking origin),(G00 velocity of each axis)]/Kp}*4 32[X-axis reasonable following error] 33[Y-axis reasonable following error] 34[Z-axis reasonable following error]		

Alarm ID	Pair-024	Alarm title	【Serious double loop position error exceeds】
description	When the instructions sent by the controller and those sent back by the second encoder exceed the range set by parameter 3817.		
Possible cause	1. Mechanism interference. 2. Motor parameters are set incorrectly. 3. Optical ruler parameters are incorrectly set. 4. Abnormal motor feedback signal or interference. 5. The optical ruler signal is abnormal or interfered. 6. The loose reading head of the optical ruler leads to the disturbance of the motor encoder.		
Elimination method	1. Check the external motion mechanism. 2. Check whether motor resolution setting Pr61~76/Pr81~96/Pr161~Pr176 is correct. 3. Check whether the optical ruler resolution Pr261~276/Pr301~Pr316 is set correctly. 4. Check whether the motor encoder works properly, or send the motor back to the wire away from the high-power electromagnetic equipment. 5. Check whether the power supply of the optical ruler is stable, or route the optical ruler away from high-power electromagnetic devices. 6. Verify that the reading head is not affected by the mechanism travel movement.		

Alarm ID	Pair-025	Alarm title	【Exceeded the forward hardware travel limit】
description	The servo motor is moving against the forward hardware travel limit switch.		
Possible cause	1. Bed move beyond protection point. 2. The hardware travel switch is damaged or disconnected. 3. Controller signal input is abnormal.		
Elimination method	1. When the alarm occurs, the bed stays on the switch. Please move the axis in the opposite direction in manual mode. 2. The bed is not on the switch when the alarm occurs. Please check the switch element, switch circuit, IO terminal block, terminal block 24V power supply. 3. The I/O card is abnormal.		

Alarm ID	Pair-033	Alarm title	【Failed to read the absolute position】
description	When the absolute encoder is used, the communication connection between the controller and the driver fails. This alarm message is issued.		

Possible cause	1. The drive is not powered on. 2. The communication cable between the controller and the driver falls off. 3. The absolute adapter plate is damaged.
Elimination method	1. Ensure that the power supply to the driver is normal. 2. Check whether the cable connection is correct. 3. Replace the absolute adapter plate.

Alarm ID	**Pair-035**	**Alarm title**	**【Absolute position battery failure】**
description	When the absolute encoder is used, the communication connection between the controller and the driver is successful, but the obtained motor encoder position is 0, the alarm message is issued.		
Possible cause	Absolute encoder battery is out of power		
Elimination method	Replace the battery		

Driver alarm -ALARM

Alarm ID	**AL-200**	**Alarm title**	**[Motor overheating]**
description	驱动器 KTY84 接口感测温度过高 驱动器 PTC130 接口感测温度过高		
Possible cause	1. Abnormal motor cooling system. 2. Digital sensing signal setting error. 3. The KTY84 analog temperature sensing setting is incorrect. 4. The fixed current of the motor is not set accurately. 5. The acceleration and deceleration time is too short. 6. Overload.		
Elimination method	1. Check the motor cooling system. 2. Modify parameters Pn-50A (P1-40) to Pn-50F (P1-61) according to the digital temperature sensing feedback mode (A or B connection). 3. Check whether the wiring is loose or abnormal, and check whether parameters Pn-740 (P1-30) and Pn-741 (P1-31) are set correctly. 4. Check the rated current parameters. 5. Check the addition and subtraction parameters to extend the acceleration and deceleration time. 6. Check whether the load rate continues to exceed 100 % and replace the motor with a larger power.		

Alarm ID	AL-201	Alarm title	【Excessive motor speed】
description	Speed feedback is greater than 120% of the maximum RPM.		
Possible cause	1. Motor power line U V W phase sequence error. 2. Encoder exception. 3. Motor parameter loading error. 4. The servo system is overshoot. 5. The speed command changes over time. 6. The drive software version is too old. 7. Encoder communication leakage packet, resulting in excessive speed jump.		
Elimination method	1. Encoder function test is carried out, whether alarm AL-302 (AL-24) is issued between tests. Refer to the manual "Automatic Tuning of New Generation Drive". 2. Perform encoder functional tests to see if any alerts are issued between tests. Refer to the manual "Automatic Tuning of New Generation Drive". 3. Modify the phase sequence of power line or modify the parameter Pn-021 (P3-22). (0 changed to 1,1 changed to 0). 4. If the polarity is correct, see the following possible causes. 5. Observe whether the driver parameter Pn-7XX is consistent with the motor nameplate parameter. 6. If the motor parameters are not consistent with the nameplate, please record the motor model and contact the technical center colleagues in Suzhou and Taiwan for assistance in providing correct motor parameters and loading. 7. The vibration of the machine was observed. Adjust the gain parameters Pn-100~Pn-102 (P2-01~P2-03). 8. Check whether the command planning of the controller is too sharp. Adjust the acceleration and deceleration time constant of the controller to a larger value. 9. Please upgrade to versions later than single-axis 1.4.12 and 4-in-1 2.0.25. 10. The velocity inching waveform is captured to observe whether the velocity change is discontinuous. 11. Check whether Pn-D73 to Pn-D76 (D1-28, D1-29, D1-46, D1-47) has a value. 12. Check whether the shielded wire is really pulled to the motor ground lock at the encoder wire pair in the junction box.		

Alarm ID	AL-202	Alarm title	[Motor overload]
description	The motor exceeds the time S2 can withstand continuous overload.		
Possible cause	1. Due to mechanical factors, the motor is blocked, resulting in excessive load during operation. 2. Continuous operation of loads exceeding driver rating. 3. The encoder and motor are connected incorrectly. 4. Encoder exception.		

Elimination method	1. Check whether there is too much difference between running command and motor speed feedback. 2. Check whether the load rate continues to exceed 100%, which can increase the motor capacity or reduce the load, or correct 3. Controller acceleration and deceleration time constant, increase its value. 3. Check the connection between the encoder and U, V, and W. Refer to the manual "Wiring and Signal" for correct connection. 4. Perform encoder functional tests to see if any alerts are issued between tests. Refer to the manual "Automatic Tuning of New Generation Drive".

Alarm ID	**AL-300**	**Alarm title**	**【Encoder failure】**
description	Encoder down, unable to correctly return position information.		
Possible cause	1. Motor overheating. 2. Noise interference. 3. Hardware failure.		
Elimination method	1. Check whether the encoder fails due to overheating of the motor. If the encoder fails, parameter Pn-90E (P3-34) should be 5. 2. Check whether the shielded wire is really pulled to the motor ground lock at the encoder wire pair in the junction box. 3. Replacement encoder.		

Alarm ID	**AL-301**	**Alarm title**	**【Encoder failure】**
description	Encoder down, unable to correctly return position information.		
Possible cause	1. Motor overheating. 2. Noise interference. 3. Hardware failure.		
Elimination method	1. Check whether the encoder fails due to overheating of the motor. If the encoder fails, parameter Pn-90E (P3-34) should be 5. 2. Check whether the shielded wire is really pulled to the motor ground lock at the encoder wire pair in the junction box. 3. Replacement encoder.		

Alarm ID	**AL-307**	**Alarm title**	**[Encoder communication anomaly 】**
description	Serial encoder communication packet error too many times, may cause the servo control abnormal		

Possible cause	1. Cue encoder communication is interfered. 2. Bad contact of encoder connector. 3. Driver end encoder connection terminal air cold welding. 4. Encoder cable shield is poorly grounded. 5. Encoder firmware exception. 6. Encoder hardware exception.
Elimination method	1. Check whether drive status variables Pn-D73 to Pn-D76 (D1-28, D1-29, D1-46, D1-47) have values. 2. Check whether the encoder, motor and machine are grounded. 3. Check whether the wiring between the encoder wire and the motor is grounded at double ends. 4. Check the connection between the encoder and the driver. 5. Check the connection between encoder and motor, welding is true. 6. Check whether drive status variables Pn-D73 to Pn-D76 (D1-28, D1-29, D1-46, D1-47) have values. 7. Check whether the metal part of the first encoder interface at the driver end is connected to the driver housing. 8. If this problem, for the driver product defects. Please contact technical center of Suzhou and Taiwan for driver replacement. 9. If the three methods above have no effect, then. 10. Both sides of the encoder wire are covered with magnetic rings. 11. As far as possible, the encoder should avoid walking with the motor power line or other large power lines. 12. Backup motor parameters, record driver and encoder versions, and update encoder firmware to V1.8.14 or above.

Alarm ID	**AL-302**	**Alarm title**	**【Excessive position error】**
description	The error between the position command and the position callback is too large.		
Possible cause	1. Torque limit is too low. 2. Excessive external load. 3. The control command changes over time. 4. The inertia of the rotor is incorrectly configured. 5. The Pn-22C (P6-41) parameter is too low.		

Elimination method	1. Check parameter Pn-70A (P3-11). 2. Check whether the load ratio consistently exceeds 100%. 3. Whether the controller command planning is too drastic. 4. Adjust the controller acceleration and deceleration time constant to increase its value. 5. Reduce external load or reassess motor capacity. 6. The inertia setting of the rotor is too low and the output current is too small, resulting in the situation that the rotor cannot be pushed, resulting in a large difference between the position feedback and the command. 7. Check parameter Pn-22C (P6-41). 8. Pn-22C has a lower bound because the bandwidth of its digital filter is at least less than one-fifth of the sampling frequency (For example, 1/10 of the PWM frequency of 5kHz corresponds to a time constant of 0.32ms, and this lower bound should be sufficient for application).

Alarm ID	**A-910**	**Alarm title**	**overload**
description	**(Warning before it becomes an overload alert (A.710 or A.720))**		
Possible cause	1. Motor connection, encoder connection bad or bad connection 2. Motor operation exceeds overload protection characteristics 3. The motor is not driven due to mechanical factors, resulting in excessive load during operation 4. Overload warning value (Pn52B) is inappropriate, 5. The servo unit is faulty		
Elimination method	1. Confirm whether the motor is connected and the encoder is connected. 2. Rediscuss load conditions and operating conditions. Or reexamine motor capacity. 3. Improve mechanical factors. 4. Properly set overload warning value (Pn52B). 5. The servo unit may be faulty. Replace the server unit. 6. Confirm the overload characteristics and operation instructions of the motor.		