M a c h i n i n g c e n t e r s e r i e s

U s e d e b u g d o c u m e n t a t i o n

Guangzhou Keyuan CNC Technology Co., Ltd

directory

directory

Note: Spindle port assignment with Axis 5 or Axis 6 or Axis 7 hardware;

1: The normal version hardware has 5 axes, and the spindle port can choose the 4th axis or the 5th axis;
2: The TI hardware version has 6 axes, the spindle port can be selected as a 4th axis or 5th axis or 6th axis;
3: The bus version hardware has 6 feed axes and 1 auxiliary axis, and the auxiliary axis is the 7th axis, which only has the function of pulse spindle and tapping;

The port selection has parameters to determine:
Number parameter 326 Spindle pulse port selection (4:4th 5 or other: 5th 6:6th 7:7th) (default 5th axis).
Bit parameter 5 0 Rigid tapping selects axis (0: 5th axis 1: 4th axis) (default 5th axis).
Bit parameter 5 2 Rigid tapping selects the 6th axis (0: invalid 1: effective).
Bit parameter 5 7 Rigid tapping selects the 7th axis (0: invalid 1: valid).
Other selections have no effect when selecting axis 7, and axis 4/5 are invalid when axis 6 is selected

statement

This manual explains as many different contents as possible, but because this manual explains as many different contents as possible, but because there are too

many possibilities involved to explain all the actions that can or cannot be carried out, what is not specifically stated in this manual may be considered unusable.

warn

Before the installation connection, programming and operation of this product, it is necessary to read in detail Before the installation connection, programming and operation of this product, you must read the manual of this product and the instruction manual of the machine tool manufacturer in detail, and carry out relevant operations in strict accordance with the requirements of the manual and the manual, otherwise it may cause damage to the product and machine tools, scrap of the workpiece and even personal injury.

note

The product functions and technical indicators (such as precision, speed, etc.) described in this manual are only for the product functions and technical indicators (such as precision, speed, etc.) described in this manual only for this product, the CNC machine tool of this product is installed, the actual functional configuration and technical performance are determined by the design of the machine tool manufacturer, and the functional configuration and technical indicators of the CNC machine tool are subject to the instruction manual of the machine tool manufacturer;

Although this system is equipped with a standard machine operation panel, the function of each button of the standard machine panel is defined by the PLC program

(ladder diagram). Please note that the functions of the keys in the machine panel of this manual are described for standard PLC programs!

***The contents of this manual are subject to change without notice.**

Chapter 1 Product Introduction

1.1 System power

1> Input: L, N (AC): AC220V from the isolation transformer;

2> Output: +V:+24V -V:0V ; (two ways in total).

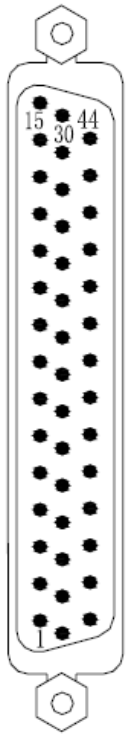
3> +24V and 0V in the system **I/O interface have the** same properties as +**24V** and **0V of the switching power supply, and can be wired straight**

+24V and 0V are brought out in the switching power supply**.**

NOTE: This power pod is for system power-on only

Chapter 2 Interface Definition and Connection

2.1 Interface CN61 input signal



Input interface(CN61)

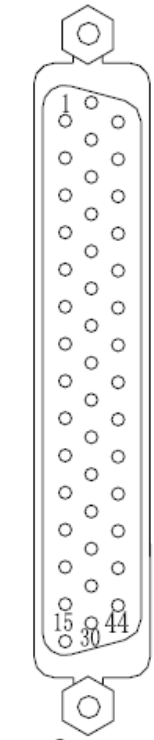
Note 1: Some input interfaces can define a variety of functions, which are **represented** by "**/" in the table;**

Note 2: When the input signal is **turned on** with **+24V, this input is valid;** When the input signal is **cut off** with **+24V, the input is invalid;**

Note 3: The +24V, .COM terminal is equivalent to the terminal of the same name of the matching power box;

Foot size	address	illustrate
17～20	dangling	
21～24	0V	Power port
25～28	+24V	Power port
1	X0.0	Protective door detection signal / external processing blowing button
2	X0.1	External pause signal (NC contact)
3	X0.2	External loosening/clamping control
4	X0.3	X-axis zero deceleration signal
5	X0.4	G31 jump signal/tool setter signal
6	X0.5	Emergency stop input signal (NC contact)
7	X0.6	Lubrication detection signal/external table control
8	X0.7	Barometric pressure detection signal
9	X1.0	A signal that the tool is released into place
10	X1.1	Tool clamping in place signal
11	X1.2	Tool changer return to zero to place signal
12	X1.3	Z-axis zero deceleration signal
13	X1.4	External cycle start (normally open contact)
14	X1.5	Spindle 1 gear in place / table clamping signal
15	X1.6	Spindle 2 in place / table release signal
16	X1.7	Tool setter overtravel signal
29	X2.0	Tool magazine forward/reverse tool in place signal
30	X2.1	Tool changer back/back in place signal
31	X2.2	Tool magazine counting signal
32	X2.3	Y-axis zero deceleration signal
33	X2.4	4-axis zero deceleration signal
34	X2.5	5-axis zero deceleration signal/disc brake signal
35	X2.6	The current cutterhead has a knife detection/disc buckle signal in place
36	X2.7	The current spindle has a knife detection/disc origin in place signal
37	X3.0	X hard limit alarm / M89K0 detection signal / macro input #1008 / external spindle magnification A
38	X3.1	Y hard limit alarm / M89K1 detection signal / macro input #1009 / external spindle magnification F
39	X3.2	Z hard limit alarm / M89K2 detection signal / macro input #1010 / external spindle magnification B
40	X3.3	4 hard limit alarm / M89K3 detection signal / macro input #1011 / external spindle magnification E
41	X3.4	External feed rate A / 5 hard limit alarm / M89K4 detection signal / macro input #1012
42	X3.5	External feed rate F/M89K5 heartbeat/macro input #1013

2.2 Interface CN62 output signal



Output interface(CN62)

Note 1: Some output interfaces can define a variety of functions, which are **represented** by "**/" in the table;**
Note 2: When the output function is active**, the output signal is internally turned on** with **0V;** When the output **function is invalid, the output signal is high impedance cutoff;**
Note 3: The +24V, .COM terminal is equivalent to the terminal of the same name of the matching power box;

Foot size	address	illustrate
19	dangling	
26～28 17～18	0V	Power port
20～25	+24V	Power port
1	Y0.0	Cooling output M08/M09
2	Y0.1	Lubrication output M32/M33
3	Y0.2	Tool clamping/loosening M16/M17
4	Y0.3	The spindle rotates M03
5	Y0.4	The spindle reverses M04
6	Y0.5	The spindle stops M05/M89 I0 output/macro output #1100
7	Y0.6	Processing blowing M07/M09
8	Y0.7	Spindle brake / M89 I1 output / macro output #1101
9	Y1.0	M41/S1 spindle gear 1/M89 I2 output/macro output #1102
10	Y1.1	M42/S2 spindle gear 2/M89 I3 output/macro output #1103
11	Y1.2	M43/S3 spindle gear 3/M89 I4 output/macro output #1104
12	Y1.3	M44/S4 spindle gear 4/M89 I5 output/macro output #1105
13	Y1.4	K1 blows M70/M71 against the knife
14	Y1.5	K2 custom output/lead screw clamping M72/M73
15	Y1.6	K4 table clamping output M10
16	Y1.7	K4 table release output M11
29	Y2.0	Guard door output M34/M35
30	Y2.1	Brake output/M89 I6 output/Macro output #1106
31	Y2.2	Tricolor light - yellow light
32	Y2.3	Tricolor light - green light
33	Y2.4	Tricolor light - red light
34	Y2.5	The second spindle stops outputting M65/M89 I7 output/macro output #1107
35	Y2.6	The second spindle reverses M64
36	Y2.7	The second spindle rotates M63
37	Y3.0	K3 custom output M74/M75
38	Y3.1	The tool magazine is rotating
39	Y3.2	Tool magazine reversal
40	Y3.3	Tool magazine forward/backward cutter

41	Y3.4	Tool magazine back/back tool
42	Y3.5	Tool magazine back indicator/disc picker output
43	Y3.6	Chip removal output M36/M37
44	Y3.7	Work light output M38/M39

2.3 I/O functions and connections

Attention!

The functional meaning of the I /O **of the** milling machine **CNC** is defined by the **PLC program (ladder diagram), when** the **milling machine** CNC **assembles the machine I/O The function is determined by the design of the machine tool factory, please refer to the manual of the machine tool manufacturer for details.**

The I/O functions in this section are described for the **standard PLC (V3.00 and above)** programs for 1 **000M series milling machines. No special instructions, please note!**

2.3.1 External pause, cycle start, emergency stop

Relevant signals

SP: External pause signal, when disconnected from +24V, pause is active;

ESP: emergency stop signal, when disconnected from +24V, emergency stop alarm;

ST: External cycle start signal, when +24V is on, the start is effective;

Signal diagnostics

Signal	SP	ESP	ST
Diagnostic address	X0.1	X0.5	X1.4
Interface pins	CN61.02	CN61.06	CN61.13
Note description	The add-in is paused	Emergency stop	Add-in start

Control parameters

Status parameter

021.0==0 External loop start (0: valid 1: invalid);

021.1==0 External pause signal (0: valid 1: invalid).

021.2==0 Emergency stop alarm signal (0: check 1: do not check).

			7	**6**	**5**	**4**	**3**	**2**	**1**	**0**
0	2	1	****	****	****	****	****	BIT2	BIT1	BIT0

BIT0 : External loop start (0: valid 1: invalid).

BIT1 : External pause signal (0: valid 1: invalid).

BIT2 : Emergency stop alarm signal (0: check 1: Do not check).

PLC parameters

K20.0==0 The external cycle start signal is (0:NO 1:NC) is valid

K20.1==1 The external pause signal is (0:NO 1:NC) is valid

K	2	0	****	****	****	****	****	****	BIT1	BIT0

BIT0: The external cycle start signal is (0:NO 1:NC) active

BIT1: The external pause signal is (0:NO 1:NC) is active

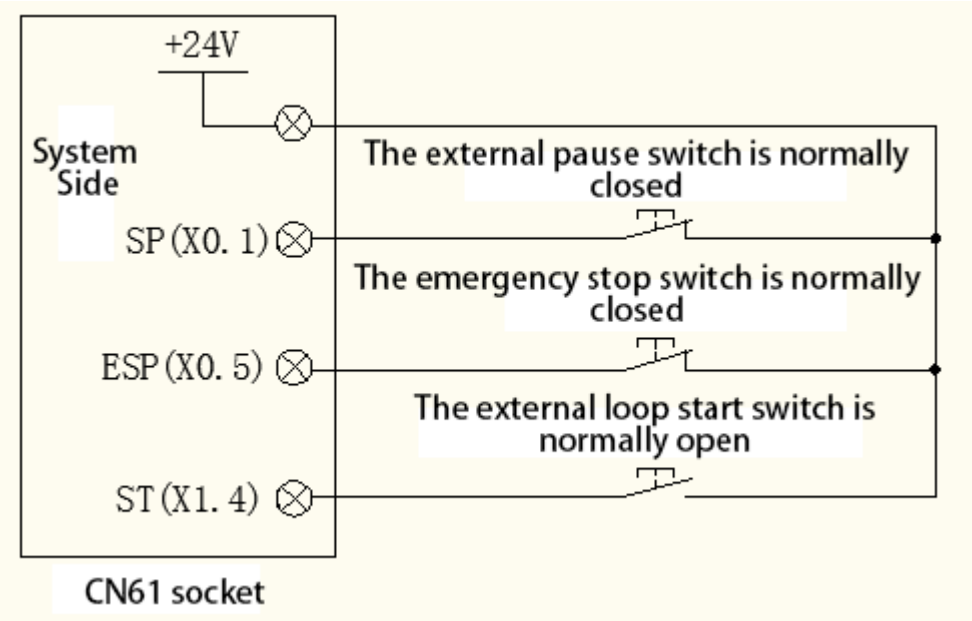
When the external suspension and cycle start are active, the external button is turned on for more than 4 seconds:

A001.6 Loop start open time is too long

A001.7 The external plug-in is paused for too long

External connection of the machine

External pause, start, emergency stop connection diagram:



2.3.2 Hard limit travel alarm and soft limit travel alarm setting

Relevant signals

LMIX: X-axis travel limit detection input

LMIY: Y-axis travel limit detection input

LMIZ: Z-axis travel limit detection input

LMI4: 4HT axis travel limit detection input

LMI5: 5HT shaft travel limit detection input

Signal diagnostics

Signal	LMIX	LMIY	LMIZ	LMI4	LMI5
Diagnostic address	X3.0	X3.1	X3.2	X3.3	X3.4
Interface pins	CN61.37	CN61.38	CN61.39	CN61.42	CN61.43
Note description	X-axis overtravel	Y-axis overtravel	Z-axis overtravel	4-axis overtravel	5-axis overtravel

remark	Address and macro input signal #1008-#1015, M89K0-K7 detection signal, external magnification switch multiplexing

Control parameters

PLC parameters

K10.5==1 Overtravel release signal for each axis (0: invalid 1: valid);

K10.6==1 The overtravel signal for each axis is (0:normally open 1:normally closed) valid

K10.7==1 Overtravel for each axis (0: invalid 1: valid);

K24.0==0 X-axis hard limit overtravel (0: valid 1: invalid);

K24.1==0 Y-axis hard limit overtravel (0: valid 1: invalid);

K24.2==1 Z-axis hard limit overtravel (0: valid 1: invalid);

K24.3==1 4-axis hard limit overtravel (0: valid 1: invalid);

K24.4==1 5-axis hard limit overtravel (0: valid 1: invalid);

K	1	0	BIT7	BIT6	BIT5	****	****	****	****	****

BIT5 : Overtravel release signal for each axis (0: invalid 1: valid).

BIT6 : The overtravel signal for each axis is (0:NO 1:NC) active

BIT7 : Overtravel for each axis (0: invalid 1: valid);

K	2	4	BIT7	BIT6	BIT5	BIT4	BIT3	BIT2	BIT1	BIT0

BIT0 : X-axis hard limit overtravel (0: valid 1: invalid);

BIT1 : Y-axis hard limit overtravel (0: valid 1: invalid);

BIT2 : Z-axis hard limit overtravel (0: valid 1: invalid);

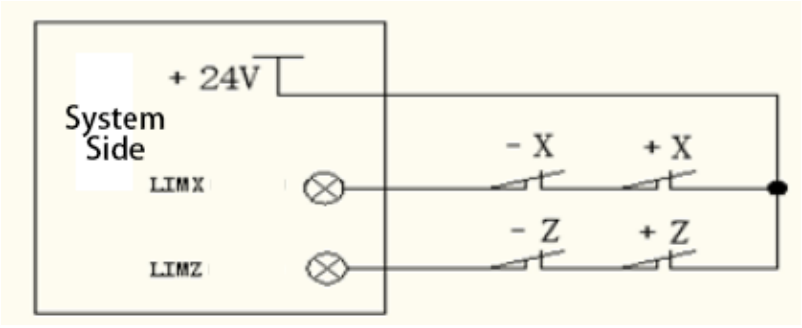
BIT3 : 4-axis hard limit overtravel (0: valid 1: invalid).

BIT4 : 5-axis hard limit overtravel (0: valid 1: invalid).

Only one limit switch is required for each axis hard limit, and the positive and negative alarm of the hard limit is judged by the axis movement direction.

External connection of the machine

Hard limit travel alarm connection diagram:



Steps to set the soft limit:

1、 Set the status parameter P22.1 to 1 (the soft limit is valid before zeroing).

2、 Then enter the maximum (small) stroke machine coordinates in the positive (negative) direction of the X axis into the data parameter P011 and data parameter P010, respectively

3、 Enter the maximum (small) stroke machine coordinates in the positive (negative) direction of the Y axis into the data parameter P013 and data parameter P012, respectively

4、 Enter the maximum (small) stroke machine coordinates in the positive (negative) direction of the Z axis into the data parameter P015 and data parameter P014, respectively

5、 INPUT THE MAXIMUM (SMALL) STROKE MACHINE COORDINATES IN THE POSITIVE (NEGATIVE) DIRECTION OF THE 4TH AXIS INTO THE DATA PARAMETER P017 and data parameter P016 respectively

6、 INPUT THE MAXIMUM (SMALL) STROKE MACHINE COORDINATES IN THE POSITIVE (NEGATIVE) DIRECTION OF THE 5TH AXIS INTO THE DATA PARAMETER P019 and data parameter P018 respectively

Status parameters:

022.1==1 Soft limit before mechanical zero (0: invalid 1: valid);
022.2==1 When issuing an overtravel command, alarm at overtravel (0:front 1:after);
022.3==0 Zero return mode soft limit (0: invalid 1: valid);

0	2	2	****	****	****	****	BIT3	BIT2	BIT1	BIT0

BIT1 : Soft limit before returning to mechanical zero (0: invalid 1: valid);
BIT2 : When the overtravel command is issued, alarm is given in the overtravel (0:front 1:after);
BIT3 : Soft limit in the zero return mode (0: invalid 1: valid);

Data parameters:

0	1	0	X-axis negative maximum travel (default: -9999.9999).
0	1	1	X-axis forward maximum stroke (default: 9999.9999).
0	1	2	Y-axis negative maximum travel (default: -9999.9999).
0	1	3	Y-axis forward maximum stroke (default: 9999.9999).

0	1	4	Z-axis negative maximum travel (default: -9999.9999).
0	1	5	Z-axis forward maximum travel (default: 9999.9999).
0	1	6	4th axis negative maximum stroke (default: -9999.9999).
0	1	7	4th axis forward maximum travel (default: 9999.9999).
0	1	8	5th axis negative maximum stroke (default: -9999.9999).
0	1	9	5th axis forward maximum travel (default: 9999.9999).

◆ **Shield each axis soft limit**

Parameter 27.0 X-axis shielded soft limit alarm (0: invalid 1: valid).

Parameter 27.1 Y-axis shield soft limit alarm (0: invalid 1: valid).

Parameter 27.2 Z-axis shielded soft limit alarm (0: invalid 1: valid).

PARAMETER 27.3 4TH AXIS SHIELD SOFT LIMIT ALARM (0: INVALID 1: VALID).

PARAMETER 27.4 5TH AXIS SHIELD SOFT LIMIT ALARM (0: INVALID 1: VALID).

Note: When set to a rotating axis, it is recommended to shield the soft limit of the axis.

Note: With 6th axis hardware interface or bus version system new 6th axis soft limit

Parameter 29.1 6th axis shield soft limit alarm (0: invalid 1: valid).

Parameter 2 9.2 6th axis shield hard limit alarm (0: invalid 1: valid).

Number parameter 435 6th axis negative maximum travel

Number parameter 436 6th axis forward maximum travel

2.3.3 The machine is returned to zero

Relevant signals

DECX: X-axis deceleration signal

DECY: Y-axis deceleration signal

DECZ: Z-axis deceleration signal

DEC4: 4-shaft deceleration signal

DEC5: 5-shaft deceleration signal

Signal diagnostics

Signal	DECX	DECZ	DECY	DEC4TH	DEC5TH
Diagnostic address	X0.3	X1.3	X2.3	X2.4	X2.5
Interface pins	CN61.04	CN61.12	CN61.32	CN61.33	CN61.34
Note description	X-axis zero deceleration signal	Z-axis zero deceleration signal	Y-axis zero deceleration signal	4-axis zero deceleration signal	5-axis zero deceleration signal; Note that the address is multiplexed with the disc tool changer brake signal

Control parameters

Status parameter

006.0==1 Zero mode selection stop (0:rear 1:front);

006.3==1 Zero return mode (0:none 1:Yes) one turn signal;

012.0==1 After powering on, before returning to the zero point of the machine, manual fast movement (0: invalid 1: valid);

026.0==0 Select the (x) axis to return to zero in the direction of (0: positive direction 1: negative direction);

026.1==0 Select the (Y) axis to return to zero in the direction of (0: positive direction 1: negative direction) back to zero;

026.2==0 Select the (Z) axis to return to zero in the direction of (0: positive direction 1: negative direction)

back to zero;

026.3==0 Select the (4) axis to return to zero in the direction of (0: positive direction 1: negative direction);

026.4==0 Select the (5) axis to return to zero in the direction of (0: positive direction 1: negative direction);

0	0	6	****	****	BIT5	****	BIT3	****	****	BIT0

BIT0 : Zero mode selection stop (0:rear 1:front).

BIT3 : Zero return mode (0:none 1:Yes) one turn signal;

0	1	2	****	****	****	****	****	****	****	BIT0

BIT0 : After powering on, before returning to the zero point of the machine, manual fast movement (0: invalid 1: valid).

0	2	6	****	****	****	BIT4	BIT3	BIT2	BIT1	BIT0

BIT0 : Select the (x) axis to return to zero in the direction of (0:positive direction 1: negative direction).

BIT1 : Select the (Y) axis to return to zero in the direction of (0:positive direction 1: negative direction).

BIT2 : Select the (Z) axis to return to zero in the direction of (0: positive direction 1: negative direction).

BIT3 : Select the (4) axis to zero as (0: positive direction 1: negative direction).

BIT4 : Select the (5) axis to return to zero in the direction of (0: positive direction 1: negative direction).

PLC parameters

K22.3==0 5TH shaft deceleration signal is (0:normally closed 1:normally open) active

K22.4==0 X-axis deceleration signal is (0:NC 1:DEAD) active

K22.5==0 Z-axis deceleration signal is (0:normally closed 1:normally open) active

The K22.6==0 Y-axis deceleration signal is (0:NC 1:NO) active

K22.7==0 4TH shaft deceleration signal is (0:NC 1:NORMALLY open) valid

K22.0==0 The Z axis is not at the zero point position Other axes (0:Yes 1:No) can return to zero

K	2	2	BIT7	BIT6	BIT5	BIT4	BIT3	****	****	****

BIT3 : 5TH shaft deceleration signal is valid for (0:NC 1:NOR).

BIT4 : X-axis deceleration signal is (0: normally closed 1: normally open) active

BIT5 : Y-axis deceleration signal is (0:NC 1:NOR) active

BIT6 : Z-axis deceleration signal is (0:NC 1:NO) active

BIT7 : 4TH AXIS DECELERATION SIGNAL IS VALID FOR (0:NC 1:NORMALLY OPEN).

Data parameters

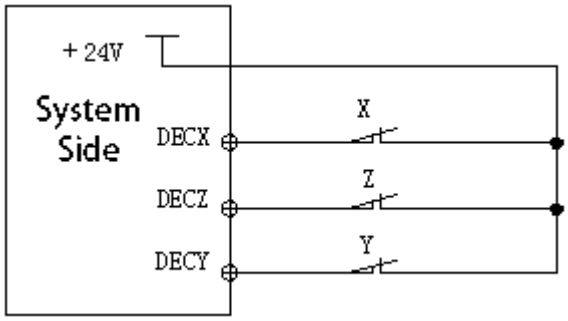
0	7	0	The high-speed speed of the X-axis back to the mechanical zero point (default: 4000).

0	7	1	The Y-axis returns to the high speed of the mechanical zero point (default: 4000).

0	7	2	The high-speed speed of the Z-axis back to the mechanical zero point (default: 4000).
0	7	3	4-axis return to mechanical zero at high speed (default: 4000).
0	7	4	5-axis return to mechanical zero speed (default: 4000).
0	8	0	The low-speed speed at which the full axis returns to the mechanical zero point (default: 40).

Note: The acceleration and deceleration time constant when returning to zero is the number parameter 1 No. 44 [non-pre-reading mode, linear time constant of acceleration and deceleration before cutting/returning to zero] generally does not need to be adjusted, and this parameter can be reduced when the deceleration distance needs to be reduced.

Diagram of the external **connection of** the machine

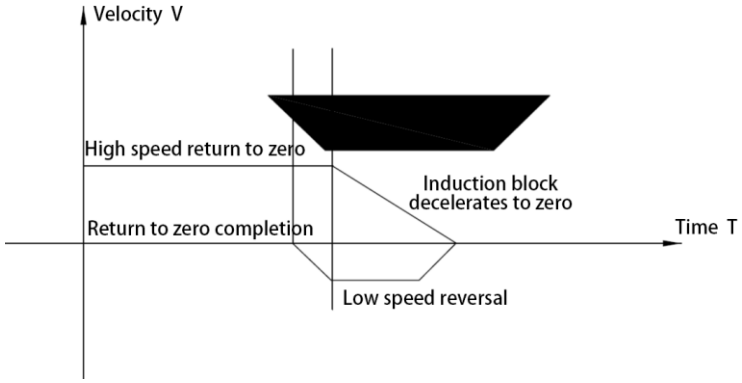


- **New back to zero control (version 2021):**
  - **Zero before the stop (bit parameter 6.0=1, it is recommended to use the stop front zero, there is no AB method before the** stop**).**

1) There is no turn signal before the stop (bit parameter 6.3=0 no turn signal).

When using the stop to zero before and selecting the zero return method without a turn signal, it is consistent with the previous version of zero return;

Description of the zero return action:

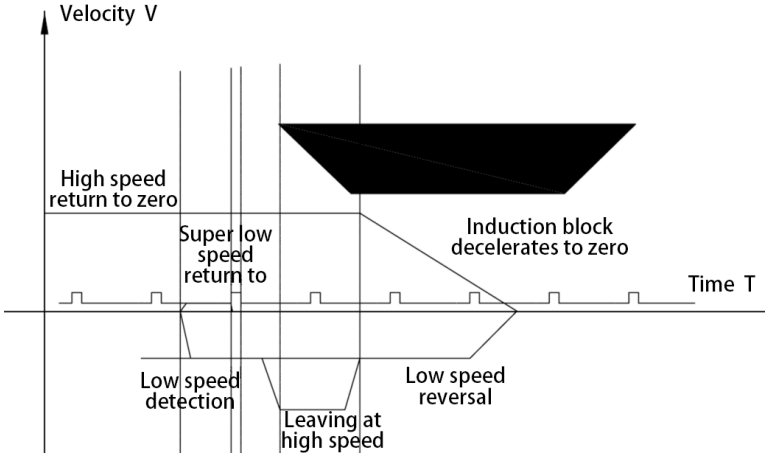


No turn signal stop before zeroing

2) There is a turn signal back to zero in front of the stop (bit parameter 6.3=1 has a turn signal).

When using the stop to zero before and selecting the zeroing method with a rotation signal, it will change from the previous version of zeroing;

Description of the zero return action:



There is a turn before the signal stop returns to zero

Description of the previous turn signal back to zero of the new stop:

New zero high-speed to find the stop, after decelerating to zero, return to zero low-speed reverse to leave the stop, after leaving the stop, move the set set to find a turn signal distance at zero high-speed speed (when the set distance is 0, do not perform the high-speed part), and then drop back to low speed to find a turn signal, find a turn signal, then move the number of pulses set, and finally reverse transfer back to 0.05 times the speed of zero low speed to detect a turn signal again, and stop after detecting a turn signal, Zero return is complete.

New parameters:

Number parameter 382 Zero return detection reserved pulse number (default 10).

The number parameter 383 X-axis returns to zero and leaves the stop to detect the signal distance of one revolution

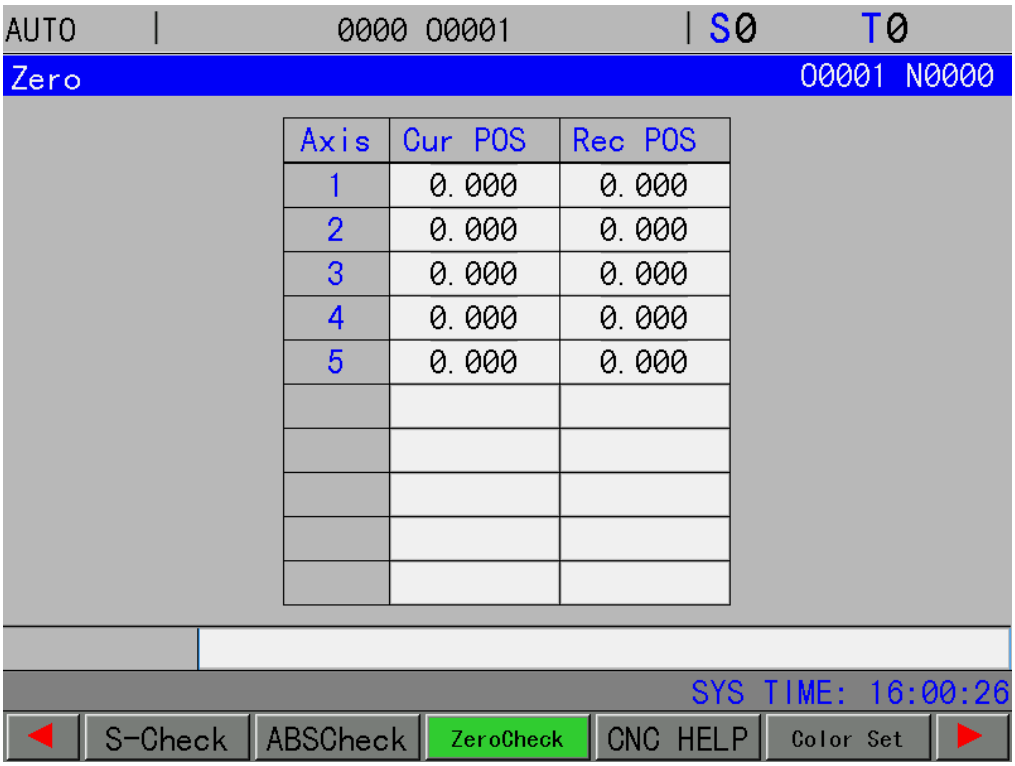
The number parameter 384 Y-axis returns to zero and leaves the stop to detect the signal distance of one revolution

The number parameter 385 Z-axis returns to zero and leaves the stop to detect the signal distance of one revolution

Number parameter 386 4-axis zero off the stop to detect a turn signal distance

Data parameter 387 5-axis zero off the stop to detect a turn signal distance

When using a turn signal to zero inaccuracy, it should generally be a critical state for the motor turn signal and the stop position, at this time you can modify the 383-387 axis to leave the stop to detect the signal distance, when the pitch is 10 can be set to 8/9, the pitch is 5 can be set 3/4 (the position of each zero point off the stop can be detected on the diagnostic < zero point detection >).



At the same time, the zero detection function can be set:

The number parameter 081 returns to zero detection range mm

Bit parameter 026.6 The zero detection function (0:No 1:Yes) is valid

bit parameter 026 7 Zero detection function (0: detection 1: self-learning).

Function usage: (only the zero mode before the stop one turn signal is valid)

When using the zero detection function, the bit parameter 2 6.6=1, first open the zero detection function self-learning bit parameter 26.7=1, set the zero error detection range parameter 8No. 1, first after the correct zeroing of each axis, Then set the zero detection function to the detection position parameter 26.7=0; After the zeroing, the zeroing accuracy of each axis will be detected, and if the set zero error parameter 8 No. 1 is exceeded, it will automatically return to zero, and if the error value is exceeded three times, the alarm will return to zero to exceed the error value.

✧ **Use a one-turn signal to return to zero as a deceleration signal (generally only used in a 1:1 rotating shaft).**

Number parameter 382 Number of reserved pulses for zero detection (default 10)

Parameter 028.0 When the X axis returns to zero (0:No 1:Yes), a turn signal is used to slow down to zero

Parameter 028.1 When the Y-axis returns to zero (0:No 1:Yes), a rotation signal is used to slow down to zero

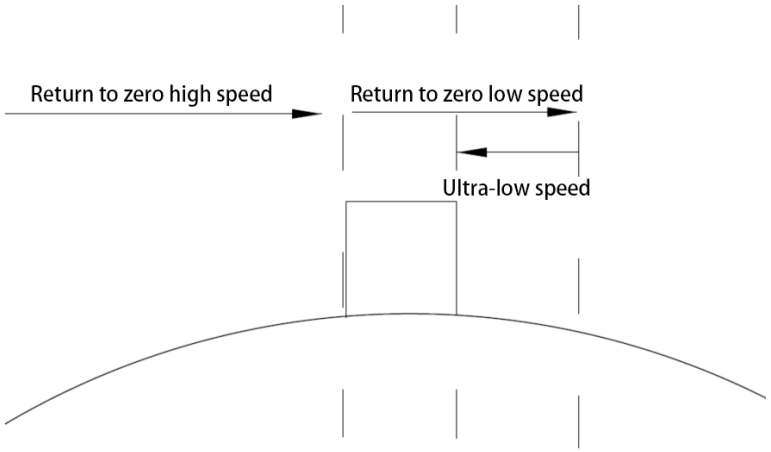
Parameter 028.2 When the Z-axis returns to zero (0:No 1:Yes), a revolution signal is used to slow down to zero

Parameter 028.3 When the 4-axis is reset to zero (0:No 1:Yes), a one-turn signal is used to slow down to zero

Parameter 028.4 When the 5-axis is returned to zero (0:no 1:yes), a one-turn signal is used to slow down to zero

When a turn signal is selected to decelerate, the zero reduction switch is not detected, and the one turn signal is directly used as the deceleration signal;

Zero back action:



Action Description:

First look for a turn signal at high speed, move the number of pulses at low speed after detecting a turn signal, and then reverse the ultra-low speed to find a turn signal to stop, and return to zero to complete.

Note: When using a rotation signal as a deceleration signal to return to zero, the zero error value is not detected;

✧ **G28 back to reference point control**

When the system default parameter machine tool has not returned to zero, when the G 28 reference point is executed, each axis is looking for the zero switch to zero at the zero speed, and after the system is zeroed, when the first reference point of the G28 is executed, each axis returns to the first reference point position at G0 speed.

Bit parameter 15.1 [G28 command when reference point is not established (0: use block 1: alarm)] (default 0).

When the machine tool is not returned to zero, but the parameter 15.1=1, command G28 command when the system alarm.

After version 2021, the software can be set to find the stop back to zero every time the G 28 command is executed;

Bit parameter 27.5 X-axis G28 back to the reference point (0:no 1:Yes) every time the stop is used

Parameter 27.6 Y-axis G28 back to the reference point (0:no 1:Yes) every time the stop is used

Parameter 27.7 Z-axis G28 back to the reference point (0:no 1:Yes) every time the stop is used

POSITION 28.5 4TH AXIS G28 BACK TO THE REFERENCE POINT (0:NO 1:YES) EVERY TIME THE STOP IS USED

POSITION PARAMETER 28.6 5TH AXIS G28 BACK TO THE REFERENCE POINT (0:NO 1:YES) EACH TIME THE STOP IS USED

✧ **The zero position offset is used**

When the machine tool needs to return to the command position, you can use the zero grid offset function, use the method: first clear the grid offset of the axis that needs to be offset (number parameter 7 5-79), set the correct gear ratio of each axis, perform the machine tool to zero, wait for the machine tool to return to zero, and move the manual / handwheel to move the axis that needs to be offset to the required position, Record the current machine coordinate value of the axis and input it into the grid offset of the axis (number parameter 7 5-79); The next time the axis returns to zero, the axis first returns to the zero position, and then moves to the offset position at the speed of zero low speed or zero high speed (bit parameter 24.6) after reaching the position, the zero return is completed, and the machine coordinates are cleared.

Bit parameter 24 The 6 return to zero grid offset speed moves at (0: zero low speed 1: each axis returns to zero high speed).

Number parameter 75 The grid offset or reference point offset of the X-axis

Number parameter 76 Grid offset or reference point offset for the Y-axis

Number parameter 7 7 The grid offset or reference point offset for the Z axis

NUMBER 78 4TH AXIS GRID OFFSET OR REFERENCE POINT OFFSET

NUMBER 79 5TH AXIS GRID OFFSET OR REFERENCE POINT OFFSET

✧ **Manually return to the reference point control after the** reference point is established **for memory**

The reference point is established (zero return is completed) and the mechanical zero point memory is set (bit parameter 15.0=1), at this time, the manual zero return can be set to the zero speed, and the speed can be selected G0 speed or the high speed of each axis to zero.

Bit parameter 15 0 Mechanical zero point (0: no memory 1: memory).

Bit parameter 15 3 After the reference point is established for memory, the reference point is manually returned to the reference point as (0:G0 fast 1:zero high-speed) speed

Note: Absolute values are set to remember zero or do not use absolute values (bit parameter 4. 0-4.4) Memory zero is also controlled

✧ **The use of the reference point can be returned without returning to the mechanical zero point**

After the absolute value system is connected to the memory zero point, it can directly return to the zero point, but in the case of not absolute value, when the zero switch is not installed or the one turn signal is not used

to slow down, the parameter parameter No. 4 and bit parameter 15 that need to be set if you want to manually return to the set mechanical zero point need to be set;

Bit parameter 4 0 The X-axis (0:Yes 1:No) reads absolute values

Bit parameter 4 1 The Y axis (0:Yes 1:No) reads absolute values

Bit parameter 4 The 2 Z-axis (0:Yes 1:No) reads absolute values

Bit parameter 4 3 The 4th axis (0:Yes 1:No) reads absolute values

Bit parameter 4 4 The 5th axis (0:Yes 1:No) reads absolute values

Bit parameter 15 0 Mechanical zero point (0: no memory 1: memory).

Bit parameter 15 3 After the reference point is established for memory, the reference point is manually returned to the reference point as (0:G0 fast 1:zero high-speed) speed

When absolute values are not used (bit parameter 6. 5=0), when setting the corresponding parameter of each axis of the bit parameter 4.0-4.4 to 1 and not reading the absolute value, set the bit parameter 1 5.0 to 1 memory zero point, at this time when manually returning to zero, the axis that does not read the absolute value will not look for the zero switch, and directly return to the zero point position of the setting machine tool, If there is no axis set that does not read absolute values, the zero-finding switch action will be performed.

Example: 3-axis XYZ, not the absolute value system (bit parameter 6.5=0) setting, set the bit parameter 4.0=0, 4 .1=0、4.2=1、15.0=1； At this time, when performing the manual zero return or return to reference point action, the X Y axis returns to zero to find the zero switch to zero, and the Z axis directly returns to the set mechanical zero point position.

Note: In the absolute value (6.5=1) setting, the bit parameter 4.0-4.4 **setting parameters mean different, is the absolute value motor shaft must be set** to **0** absolute value function to **be effective, do not set the axis of the absolute value function to 1 When it is set to zero after setting to 1, you need to find the zero switch to return to zero.**

Example: 3-axis XYZ, absolute value system (6.5=1) setting, set the bit parameters 4.0=0, 4.1=0 、4.2=1、15.0=1； At this time, when performing the manual zero return or return to the reference point action, the X Y axis directly returns to the set mechanical zero point position, and the Z axis returns to zero to find the zero switch to zero.

Note: With 6th axis hardware interface or bus version system, new 6th axis zero return X4.0 6th axis zero deceleration signal

Parameter 26.5 The 6th axis returns to the machine in the direction of zero (0: positive direction 1: opposite direction).

Parameter 28.7 6th axis G28 back reference point (0:no 1:yes) every time the stop is used

Parameter 29.0 When the 6th axis returns to zero (0:No 1:Yes), a one-turn signal is used to slow down to zero

Number parameter 388 6 axis zero off the stop to detect a turn signal distance

NUMBER PARAMETER 448 6TH AXIS RETURN TO ZERO HIGH-SPEED SPEED
NUMBER PARAMETER 450 6TH AXIS GRID OFFSET OR REFERENCE POINT OFFSET

2.3.4 Cooling pump control

Relevant command signals (defined by standard PLC programs).

Signal type	symbol	Signal interface	address	Function description	remark
Output signal	M08	CN62.1	Y0.0	Cooling output	
Instruction format	M08			Coolant on	
	M09			Coolant off	

PLC parameters

K10.1==0 spindle/lubrication/cooling/blowing output at the end of the program (0: off 1: hold);

Note: When resetting, the spindle/lubrication/cooling/blowing output must be disconnected.

K	1	0	****	****	****	****	****	****	BIT1	****

BIT1 : Spindle / lubrication / cooling / blowing output at the end of the program (0: off 1: hold) ;

PLC parameters

T	0	5	M code execution duration (ms) (default: 1).

Function description (standard PLC program definition).

After the CNC is powered on, M09 is valid, that is, the M08 output is invalid. M08 is executed, M08 output is effective, cooling pump open; Execute M09, cancel M08 output, and turn off the cooling pump.

2.3.5 Processing blowing control

Relevant command signals (defined by standard PLC programs).

Signal type	symbol	Signal interface	address	Function description	remark
Output signal	M07	CN62.07	Y0.6	Blowing output	
Instruction format	M07			Blow on the air	
	M09			Blow off	
Input signal		CN61.01	X0.0	External blow button	K20.5==0

PLC parameters

K10.1==0 spindle/lubrication/cooling/blowing output at the end of the program (0: off 1: hold);

K20.5==0 (0: external processing blowing button 1: protective door signal) is valid;

Note: When resetting, the spindle/lubrication/cooling/blowing output must be disconnected.

K	1	0	****	****	****	****	****	****	BIT1	****

BIT1 : Spindle / lubrication / cooling / blowing output at the end of the program (0: off 1: hold) ;

K	2	0	****	****	BIT5	****	****	****	BIT1	****

BIT5 : (0: External processing blowing button 1: protective door signal) Effective;

PLC parameters

T	0	5	M code execution duration (ms) (default: 1).

Function description (standard PLC program definition).

After the CNC is powered on, M09 is valid, that is, the M07 output is invalid. Perform M07, M07 output is effective, processing blowing open; Perform M09, cancel M07 output, process blowing off.

2.3.6 Lubrication control

Relevant command signals (defined by standard PLC programs).

Signal type	symbol	Signal interface	address	Function description	remark
Output signal	M32	CN62.2	Y0.1	Lubricating output	
Instruction format	M32			Lubricate on	
	M33			Lubricating off	

Control parameters

PLC parameters

K10.1==0 spindle/lubrication/cooling/blowing output at the end of the program (0: off 1: hold);

K16.2==1 Turn on when automatic lubrication is effective (0:no 1:Yes) output lubrication;

Note: When resetting, the spindle/lubrication/cooling/blowing output must be disconnected.

K	1	0	****	****	****	****	****	****	BIT1	****

BIT1 : Spindle / lubrication / cooling / blowing output at the end of the program (0: off 1: hold) ;

K	1	6	****	****	****	****	****	BIT2	****	****

BIT2 : Turn on when automatic lubrication is effective (0:no 1:Yes) output lubrication;

T	0	5	M code execution duration (ms) (default: 1).

T	1	3	Automatic lubrication output time (0~65535ms).

T	5	3	Automatic lubrication interval time (0~65535s) unit: seconds

- Feature description

1) When DT13 is not zero, automatic lubrication is effective, and the lubrication will be output after the time set by DT 53 every interval (the time set by DT 13); Manual and command can also turn on or off the lubrication output at the time set by DT13 (the timer will restart after this operation);
2) When automatic lubrication is effective, if K16.2 is set to 1, the system will output the lubrication as soon as it is powered on, and the output time is the time set by DT13, and if K16.2 is set to 0, the lubrication will only start after the time set by DT 53 after power-on.
3) When DT1 3=0, manual lubrication is effective, if the parameter DT13=0, the output will not be automatically turned off, if not 0, the output time is reached DT13 will automatically turn off the lubrication output after the time set;
4) When DT53=0, the lubrication output is closed after entering DT13, and the output is no longer spaced; When DT53 is not 0, the lubrication output is closed after entering DT13, and the output is in the output after the DT53 interval output time;

2.3.7 Lubrication testing

Lubrication detection signal diagnostics

Diagnostic address	X0.6
Interface pins	CN61.07
remark	Function multiplexing option K20.6=0

PLC parameters

K16.1==0 lubrication detection signal is (0:NO 1:NC) active;

K20.6==0 X0.6 signal (0: lubrication detection 1: external table control) is valid

T	0	3	Insufficient lubricant detection delay Unit: milliseconds

Function description: In the detection time of T 03, when the lubricating fluid deficiency signal is detected, the system will issue warning signal A020.1 [insufficient lubricating oil warning, detection delay T03]; Does not affect normal processing.

2.3.8 Insufficient air pressure detection

Insufficient air pressure detection signal diagnosis

Diagnostic address	X0. 7
Interface pins	CN61.08
remark	

PLC parameters

K20.2==0 The insufficient pressure detection signal is (0:normally open 1:normally closed) is valid;

T	0	2	Insufficient air pressure detection delay Unit: milliseconds

Function description: In the T03 time detection time, when the insufficient air pressure signal is detected, the system will send an alarm signal A008 2 [Air pressure insufficient alarm, level selection K20.2 detection delay T02], note that the system will shield the signal detection during the automatic tool change, and resume the detection after the automatic tool change is completed.

2.3.9 Tricolor lamps

Relevant signals (defined by standard PLC programs).

CLPY: Tricolor light - yellow light

CLPG: Tricolor - Green light

CLPR: Tricolor - Red light

Signal diagnostics

Signal	CLPY	CLPG	CLPR
Diagnostic address	Y2.2	Y2.3	Y2.4
Interface pins	CN62.31	CN62.32	CN62.33
Note description	Yellow light	green light	red light
remark	normal	Running status	Alarm status

2.3.10 Macro variable input and output

The address relationship between the macro variables **#1008~#1015 and the input port X**

Relevant signals (defined by standard PLC programs).

Macro variables	#1015	#1014	#1013	#1012	#1011	#1010	#1009	#1008
Diagnostic address	X3.7	X3.6	X3.5	X3.4	X3.3	X3.2	X3.1	X3.0
Interface pins	CN61.44	CN61.43	CN61.42	CN61.41	CN61.40	CN61.39	CN61.38	CN61.37
remark	Macro input address and each axis hard limit overtravel, M89K0-K7 detection signal, external magnification switch, pay attention to the selection and use							

In the V2.09 version ladder diagram, you can select the input macro #1008-**#1015 ports, and the input signal can be selected as normally open/normally closed**

K034. 0 M89/macro #1000-#1007 Detection signal selection (0:X3 1:X2)

K34.0=0 Select 0 (X3.0-X3.7).

K34.0=1 Select 1 (X2.0-X2.7).

K035.0 M89K0/Macro #1000 heartbeat (0:NO 1:NC).

K035.1 M89K1/Macro #1001 heartbeat (0: normally open 1: normally closed).

K035.2 M89K2/macro #1002 heartbeat (0: normally open 1: normally closed).

K035.3 M89K3/macro #1003 heartbeat (0:NO 1:NC).

K035.4 M89K4/macro #1004 heartbeat (0:normally open 1:normally closed).

K035.5 M89K5/macro #1005 heartbeat (0:NO 1:NC).

K035.6 M89K6/macro #1006 heartbeat (0:NO 1:NC).

K035.7 M89K7/macro #1007 heartbeat (0:NO 1:NC).

The address relationship between macro variable #11 00~#1107 and output port Y

Relevant signals (defined by standard PLC programs).

Macro variables	#1107	#1106	#1105	#1104	#1103	#1102	#1101	#1100
Diagnostic address	Y2.5	Y2.1	Y1.3	Y1.2	Y1.1	Y1.0	Y0.7	Y0.5
Interface pins	CN62.34	CN62.30	CN62.12	CN62.11	CN62.10	CN62.09	CN62.08	CN62.06
remark	The macro output address and M89I0-I7 output signal, spindle stop, spindle brake, holding brake, spindle shift output multiplex, pay attention to the selection and use.							

Control parameters

PLC parameters

K028.0==1 (0: spindle stop 1: M89 I0 custom/macro output #1100) Output Y0.5 is valid

K028.1==1 (0: Spindle brake 1: M89 I1 custom/macro output #1101) Output Y0.7 is valid

K028.2==1 (0: spindle 1 gear 1: M89 I2 custom/macro output #1102) Output Y1.0 is valid

K028.3==1 (0: spindle 2 gears 1 : M89 I3 custom/macro output #1103) Output Y1.1 is valid

K028.4==1 (0: spindle 3 gears 1: M89 I4 custom/macro output #1104) Output Y1.2 is valid

K028.5==1 (0: spindle 4 gears 1: M89 I5 custom/macro output #1105) Output Y1.3 is valid

K028.6==1 (0: Brake output 1 : M89 I6 custom/macro output #1106) Output Y2.1 is valid
K028.7==1 (0: 2nd spindle stop 1: M89 I7 custom/macro output #1107) Output Y2.5 is valid
K027.0==1 (0:M89I0Custom 1:Macro output #1100)Output Y0.5 is valid
K027.1==1 (0:M89I1Custom 1:Macro output #1101)Output Y0.7 is valid
K027.2==1 (0:M89I2Custom 1:Macro output #1102)Output Y1.0 is valid
K027.3==1 (0:M89I3Custom 1:Macro output #1103)Output Y1.1 is valid
K027.4==1 (0:M89I4Custom 1:Macro output #1104)Output Y1.2 is valid
K027.5==1 (0:M89I5Custom 1:Macro output #1105)Output Y1.3 is valid
K027.6==1 (0:M89I6Custom 1:Macro output #1106)Output Y2.1 is valid
K027.7==1 (0:M89I7Custom 1:Macro output #1107)Output Y2.5 is valid

K	2	7	BIT7	BIT6	BIT5	BIT4	BIT3	BIT2	BIT1	BIT0

BIT0 : (0:M89I0Custom 1:Macro output #1100) Output Y0.5 valid
BIT1 : (0:M89I1Custom 1:Macro output #1101) Output Y0.7 valid
BIT2 : (0:M89I2Custom 1:Macro output #1102) Output Y1.0 is valid
BIT3 : (0:M89I3Custom 1:Macro output #1103) Output Y1.1 is valid
BIT4 : (0:M89I4Custom 1:Macro output #1104)Output Y1.2 is valid
BIT5 : (0:M89I5Custom 1:Macro output #1105) Output Y1.3 is valid
BIT6 : (0:M89I6Custom 1:Macro output #1106) Output Y2.1 is valid
BIT7 : (0:M89I7Custom 1:Macro output #1107) Output Y2.5 valid

K	2	8	BIT7	BIT6	BIT5	BIT4	BIT3	BIT2	BIT1	BIT0

BIT0 : (0: Spindle Stop 1: M89 I0 Custom/Macro Output #1100) Output Y0.5 is valid
BIT1 : (0: Spindle brake 1: M89 I1 custom/macro output #1101) Output Y0.7 is valid
BIT2 : (0: spindle 1 stop 1: M89 I2 custom/macro output #1102) Output Y1.0 is valid
BIT3 : (0: spindle 2 gears 1 : M89 I3 custom/macro output #1103) Output Y1.1 is valid
BIT4 : (0: spindle 3 gears 1: M89 I4 custom/macro output #1104) Output Y1.2 is valid
BIT5 : (0: spindle 4 gears 1: M89 I5 custom/macro output #1105) Output Y1.3 valid
BIT6 : (0: Brake output 1 : M89 I6 custom/macro output #1106) Output Y2.1 is valid
BIT7 : (0: 2nd spindle stop 1: M89 I7 custom/macro output #1107) Output Y2.5 valid

- **Function description:**

Program Editor:

G65 H81 P50 Q#1008 R1;//Program jumps to N50 segment when #1008=1, #1008=1 when X3.0 is switched on with +24V;
N50 G00 X50 Z50;
G65 H01 P#1100 Q1;// #1100=1 outputs Y0.5, when #1100=1 outputs the corresponding Y address;
N60 G00 X0 Z0;
G65 H01 P#1100 Q0;// #1100=0 closes output Y0.5, when #1100=0 closes the corresponding Y address;

2.3.11 M89 custom detection output

- **Programming format description:**

① M89 K_ J_ R_ P_ I_ Q_

K_: The heartbeat selects K0—K7

J_ : Wait detection time in milliseconds

R_ : Repeat output interval in milliseconds

P_ : Number of repeated outputs The range P0 is output 1 time, P1 is 2 repetitions, P2 is 3 repetitions, and so on

I_ : Select to open the output address I0—I7

Q_ : Select to turn off the output address Q0—Q7

Note: The output addresses of I/Q on and off correspond one-to-one.

M89 output I_, wait for detection K_, no detection within J_ time K_ turn off the output I_, turn on the output after R_ time has elapsed I_ alarm when no K_ is detected within the P_number of cycles, and turn off the output at the same time;

If the M89 code is completed within P_ of the K_ is detected within the number of cycles, the Q_ output is turned off at the same time, and if no Q_ is filled in, it remains I_;

If I_=Q_, turn off the I_ output when M89 is complete; If I_ ≠Q_, keep the I_ output and turn off the Q_ output.

② M89 K_ J_

When M89 does not detect K_ within J_ waiting time, an alarm is generated;

When M89 waits for K_ to be detected within J_ time, the M89 code is complete;

③ M89 I_

M89 output I_ hold

④ M89 Q _

M89 shutdown output Q_

⑤ M89 I_ J_ Q_

M89 output I_ elapsed time J_ Turn off the output Q_;

If the Q_ is not filled in, the M89 output I_ elapsed time J_ turns off the output;

If the elapsed time is I_ = Q_J_ turns off the output;

If I_ ≠Q_, keep the I_ output after elapsed time J_ and turn off the Q_ output.

⑥ M89 K_

When M89 waits for a K_ to be detected, the M89 code completes;

⑦ M89 K_ J_ I_

M89 output I_ wait for K_ to be detected, K_ detected within time J_, M89 code complete;

If no K_ is detected within the time J_, the system generates an alarm;

When no J_ or J=0 is written, it waits for the detection K_ signal until the K_ signal is detected, which completes the M89 code and generates an alarm;

⑧ M89 K_ J_ Q _

M89 closes output Q_ waits for K_ to be detected, K_ detected within time J_, M89 code completes;

If no K_ is detected within the time J_, the system generates an alarm;

When no J_ or J=0 is written, it waits for the detection K_ signal until the K_ signal is detected, which completes the M89 code and generates an alarm;

M89 L_ Minimum Hold Time for Signal Detection When the M89 K detection function is turned on, if there is no L programming, the code will be completed as soon as there is a signal, and when there is L, the code can only be completed when the signal is greater than the L editing time;

All edited formats with K can use L to detect duration.

Note: When K0-K7 is not written after M89**, the M89 code is completed directly;**

The address relationship between the M89K0—K7 heartbeat and the input port X

Relevant signal (defined by standard PLC program) [Detection signal must use normally open sensor or normally open stroke switch]

M89	K7	K6	K5	K4	K3	K2	K1	K0
Diagnostic address	X3.7	X3.6	X3.5	X3.4	X3.3	X3.2	X3.1	X3.0
Interface pins	CN61.44	CN61.43	CN61.42	CN61.41	CN61.40	CN61.39	CN61.38	CN61.37
remark	M89K0-K7 detection signal and each axis hard limit overtravel, macro input, external spindle magnification switch, 5-axis zero signal multiplexing, please pay attention to the choice of use							

In the V3.09 ladder diagram, you can select the input **M89K0-K7 ports, and the input signal can be selected as normally open/normally closed**

K034. 0 M89/macro #1000-#1007 Detection signal selection (0:X3 1:X2)

K34.0=0 Select 0 (X3.0-X3.7).

K34.0=1 Select 1 (X2.0-X2.7).

K035.0 M89K0/Macro #1000 heartbeat (0:NO 1:NC).

K035.1 M89K1/Macro #1001 heartbeat (0: normally open 1: normally closed).

K035.2 M89K2/macro #1002 heartbeat (0: normally open 1: normally closed).

K035.3 M89K3/macro #1003 heartbeat (0:NO 1:NC).

K035.4 M89K4/macro #1004 heartbeat (0:normally open 1:normally closed).

K035.5 M89K5/macro #1005 heartbeat (0:NO 1:NC).

K035.6 M89K6/macro #1006 heartbeat (0:NO 1:NC).

K035.7 M89K7/macro #1007 heartbeat (0:NO 1:NC).

The address relationship between the M89I0—**I7/Q0—Q7 output/shutdown signal and the output port Y**

Relevant signals (defined by standard PLC programs)

M89	I7/Q7	I6/Q6	I5/Q5	I4/Q4	I3/Q3	I2/Q2	I1/Q1	I0/Q0
Diagnostic address	Y2.5	Y2.1	Y1.3	Y1.2	Y1.1	Y1.0	Y0.7	Y0.5
Interface pins	CN62.34	CN62.30	CN62.12	CN62.11	CN62.10	CN62.09	CN62.08	CN62.06
remark	M89I0-I7 output signal address and macro output, spindle stop, spindle brake, holding brake, spindle shift output multiplexing, please pay attention to choose to use.							

Note: When the mechanical zero mode or program is running, the output of I0-I7/Q0-Q7 can be manually controlled **or turned off;**

The corresponding button of control I0-I7/Q0-Q7 is the number **0-7 of the PLC interface to expand the control interface.**

Control parameters

PLC parameters

K029.0==0 Disconnect the M89 I0 output when reset (0:Yes 1:No).
K029.1==0 Disconnect the M89 I1 output when reset (0:Yes 1:No).
K029.2==0 disconnects the M89 I2 output when reset (0:Yes 1:No).
K029.3==0 Disconnect the M89 I3 output when reset (0:Yes 1:No).
K029.4==0 Disconnect the M89 I4 output when reset (0:Yes 1:No).
K029.5==0 Disconnect the M89 I5 output when reset (0:Yes 1:No).
K029.6==0 Disconnect the M89 I6 output when reset (0:Yes 1:No).
K029.7==0 Disconnect the M89 I7 output when reset (0:Yes 1:No).
K028.0==1 (0: spindle stop 1: M89 I0 custom/macro output #1100) Output Y0.5 is valid
K028.1==1 (0: Spindle brake 1: M89 I1 custom/macro output #1101) Output Y0.7 is valid
K028.2==1 (0: spindle 1 gear 1: M89 I2 custom/macro output #1102) Output Y1.0 is valid
K028.3==1 (0: spindle 2 gears 1 : M89 I3 custom/macro output #1103) Output Y1.1 is valid
K028.4==1 (0: spindle 3 gears 1: M89 I4 custom/macro output #1104) Output Y1.2 is valid
K028.5==1 (0: spindle 4 gears 1: M89 I5 custom/macro output #1105) Output Y1.3 is valid
K028.6==1 (0: Brake output 1 : M89 I6 custom/macro output #1106) Output Y2.1 is valid
K028.7==1 (0: 2nd spindle stop 1: M89 I7 custom/macro output #1107) Output Y2.5 is valid
K027.0==0 (0:M89I0Custom 1:Macro output #1100)Output Y0.5 is valid
K027.1==0 (0:M89I1Custom 1:Macro output #1101)Output Y0.7 is valid
K027.2==0 (0:M89I2Custom 1:Macro output #1102)Output Y1.0 is valid
K027.3==0 (0:M89I3Custom 1:Macro output #1103)Output Y1.1 is valid
K027.4==0 (0:M89I4Custom 1:Macro output #1104)Output Y1.2 is valid
K027.5==0 (0:M89I5Custom 1:Macro output #1105)Output Y1.3 is valid
K027.6==0 (0:M89I6Custom 1:Macro output #1106)Output Y2.1 is valid
K027.7==0 (0:M89I7Custom 1:Macro output #1107)Output Y2.5 is valid

K	2	9

BIT7	BIT6	BIT5	BIT4	BIT3	BIT2	BIT1	BIT0

BIT0 : Disconnect the M89 I0 output on reset (0:Yes 1:No).
BIT1 : Disconnect the M89 I1 output on reset (0:Yes 1:No).

BIT2 : Disconnects the M89 I2 output on reset (0:Yes 1:No).
BIT3 : Disconnects the M89 I3 output on reset (0:Yes 1:No).
BIT4 : Disconnects the M89 I4 output on reset (0:Yes 1:No).
BIT5 : Disconnect the M89 I5 output on reset (0:Yes 1:No).
BIT6 : Disconnects the M89 I6 output on reset (0:Yes 1:No).
BIT7 : Disconnects the M89 I7 output on reset (0:Yes 1:No).

K	2	7	BIT7	BIT6	BIT5	BIT4	BIT3	BIT2	BIT1	BIT0

BIT0 : (0:M89I0Custom 1:Macro output #1100) Output Y0.5 valid
BIT1 : (0:M89I1Custom 1:Macro output #1101) Output Y0.7 valid
BIT2 : (0:M89I2Custom 1:Macro output #1102) Output Y1.0 is valid
BIT3 : (0:M89I3Custom 1:Macro output #1103) Output Y1.1 is valid
BIT4 : (0:M89I4Custom 1:Macro output #1104)Output Y1.2 is valid
BIT5 : (0:M89I5Custom 1:Macro output #1105) Output Y1.3 is valid
BIT6 : (0:M89I6Custom 1:Macro output #1106) Output Y2.1 is valid
BIT7 : (0:M89I7Custom 1:Macro output #1107) Output Y2.5 valid

K	2	8	BIT7	BIT6	BIT5	BIT4	BIT3	BIT2	BIT1	BIT0

BIT0 : (0: Spindle Stop 1: M89 I0 Custom/Macro Output #1100) Output Y0.5 is valid
BIT1 : (0: Spindle brake 1: M89 I1 custom/macro output #1101) Output Y0.7 is valid
BIT2 : (0: spindle 1 stop 1: M89 I2 custom/macro output #1102) Output Y1.0 is valid
BIT3 : (0: spindle 2 gears 1 : M89 I3 custom/macro output #1103) Output Y1.1 is valid
BIT4 : (0: spindle 3 gears 1: M89 I4 custom/macro output #1104) Output Y1.2 is valid
BIT5 : (0: spindle 4 gears 1: M89 I5 custom/macro output #1105) Output Y1.3 valid
BIT6 : (0: Brake output 1 : M89 I6 custom/macro output #1106) Output Y2.1 is valid
BIT7 : (0: 2nd spindle stop 1: M89 I7 custom/macro output #1107) Output Y2.5 valid

2.3.12 M90, M91 program segment loop instructions

- **Programming format description:**

  M91 C_;// Loop program segment starts, C_ number of cycles
  ;
  ;
  M90;// End of loop segment

  Example:
  T01 M06;
  M3 S1000;
  G0 X32 Z5;
  M91 C5; The loop program segment starts, 5 times (loop N10-N20 program segments).
  N10 G01 X30 Z0 F500;
  G01 Z-20;
  G01 X20 Z-40;
  G0 X32;

N20 Z0;
M90; End of the loop segment

2.3.13 M92, M93 program jump

M92 conditional jump instruction:

- Programming Format:

M92 K_ N_ ;
M92 L_ N_ ;

K_ Input high; (The range K0--K7 corresponds to #1008--#1015=1, #1008--#1015 对应输入信号参考 2.3.10)
L_ Input low; (Range L0--L7 corresponds to #1008--#1015=0)
N_ Jump program segment number;
When the conditions are met, jump to the N_ paragraph;

Example:
M92 K0 N10;//When #1008=1(X3.0=1), the program jumps to segment N10; The G65 H81 P10 Q#1008 R1 instructions are the same
G0 X50 ;
N10 G0 X30 ;
G0 X0;
M92 L0 N10;// When #1008=0(X3.0=0), the program jumps to N10 segment G65 H81 P10 Q#1008 R0 with the same instructions

M93 Unconditional jump instruction:

- Programming Format:

M93 N_ ;
N_ Jump program segment number;
Jump to N_ when running to M93 N_;

Example:
M93 N10;//Jump to N10 segment
G0 X50;
N10 G01 X30;
When running to the M93 N10 segment, the program jumps to the N10 segment and does not execute G0 X50.

2.3.14 User-defined output

Relevant signals (defined by standard PLC programs).

Diagnostic address	Y2.0	Y3. 6	Y3. 7	Y1.4	Y1.5	Y3.0
Interface pins	CN62.29	CN62.43	CN62.44	CN62.13	CN62.14	CN62.37
Note description	Guard door output M34/M35	Chip removal output M36/M37	Work light output M38/M39	Blow K1 on the knife to output M70/M71	Custom K2 output M72/M73	Custom K3 output M74/M75
remark						

PLC parameters

K017.0==0 Automatic output of work lights when the system is powered on (0: invalid 1: valid).

T	0	5	M code execution duration (ms) (default: 1).

NOTE: The reset disconnect output can be set

K36.0 Reset button (0:No 1:Yes) disconnects the guard gate output M34

K36.1 Reset button (0:No 1:Yes) disconnects the chip conveyor output M36

K36.2 Reset button (0:No 1:Yes) Disconnect the work lamp output M38

K36.3 Reset button (0:No 1:Yes) disconnects the blowing output M70 to the knife

K36.4 Reset button (0:No 1:Yes) disconnects K2 output M72

K36.5 Reset button (0:No 1:Yes) disconnects K3 output M74

2.3.15 External feed rate switch

Signal diagnostics

Diagnostic address	X3.4	X3.5	X3.6	X3.7
Interface pins	CN61.41	CN61.42	CN61.43	CN61.44
Note description	External magnification switch A	External magnification switch F	External magnification switch B	External magnification switch E
remark	The external magnification switch signal is repeated with the 5th axis hard limit address, pay attention to the selection and use			

Control parameters

PLC parameters

K010.3==0 feed rate (0: magnification can be adjusted 1: fixed at 100%)

K016.0==1 External feed multiplier switch (0: invalid 1: valid).

K	1	0	****	****	****	****	BIT3	****	****	****

BIT3 : Feed rate (0: magnification can be adjusted 1: fixed at 100%)

K	1	6	****	****	****	****	****	****	****	BIT0

BIT0 : External feed multiplier switch (0: invalid 1: valid).

- Feature description
  1) When K10.3 is set to 1, the cutting feed ratio is fixed at 100%, and the feed rate adjustment is invalid.
  2) The adjustment buttons on the panel will have no effect when an external feed rate switch is selected
  3) When tapping, the feed multiplier cannot be changed.

2.3.16 External spindle magnification switch

Signal diagnostics

Diagnostic address	X3.0	X3.1	X3.2	X3.3
Interface pins	CN61.37	CN61.38	CN61.39	CN61.40
Note description	External spindle switch A	External spindle switch F	External spindle switch B	External spindle switch E
remark	External spindle magnification switch address and M89K4-K7 detection signal, shaft overtravel signal, please pay attention to choose to use			

Control parameters

PLC parameters

K017.5==1 External magnification switch for spindle (0: invalid 1: valid).

K	1	7	****	****	BIT5	****	****	****	****	****

BIT5 : Spindle external magnification switch (0: invalid 1: valid).

- **Feature description**
  1) The adjustment buttons on the panel will have no effect when an external feed rate switch is selected
  2) When tapping, the feed multiplier cannot be changed.

2.3.17 G31 jump function / Z direction automatic tool setting function

Signal diagnostics

Diagnostic address	X0.4	X1.7
Interface pins	CN61.36	CN61.17
Note description	G31 jump function	Tool setter overtravel signal
remark	K24.7 selects the sense level	K24. 6Select the sensing level

Relevant parameters

K024.7==0 G31 jump signal is (0:normally closed 1:normally open) is valid

K024.6==0 The tool setter overtravel signal is (0:normally closed 1:normally open) valid

K	2	4

BIT7	BIT6	BIT5	****	****	****	****	****

BIT7 : G31 jump signal is (0:NC 1:LESS) active

BIT6 : The tool setter overtravel signal is (0:normally closed 1:normally open) valid

- **Function description**

1) Code format: G31 X_ Y_ Z_ F_;
2) Code function: During the execution of the code, if an external jump signal (X0.4) is entered, the execution of the code is interrupted and the next program segment is executed. This function can be used for dynamic measurement of workpiece dimensions (e.g. grinding machines), tool measurement, etc.
3) Code description: Non-modal G-code (00 group);
4) It is consistent with the G01 code address format and is similar to the use;
5) The tip radius compensation needs to be revoked before using the code;
6) In order to ensure the accuracy of the stop position, the feed speed should not be set too large.
7) Lines for subsequent segments when the jump occurs:
8) The next segment of the G31 is incremental coordinate programming, see Figure 3-13

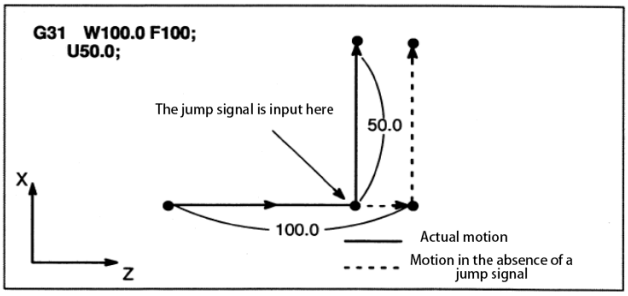


Figure 3-13

9) The next program segment of G31 is the programming of absolute coordinates of 1 axis, see Figure 3-14

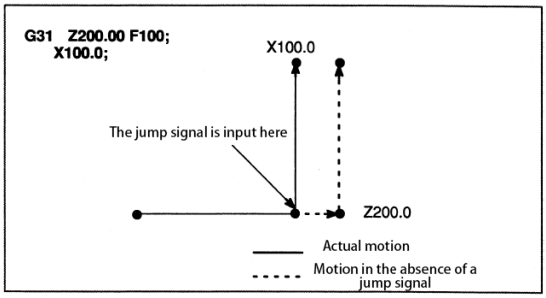


Figure 3-14

10) The next program segment of the G31 is the programming of absolute coordinates of the 2 axes , see Figure 3-15

Program: G31 Z200 F100

G01 X100 Z300

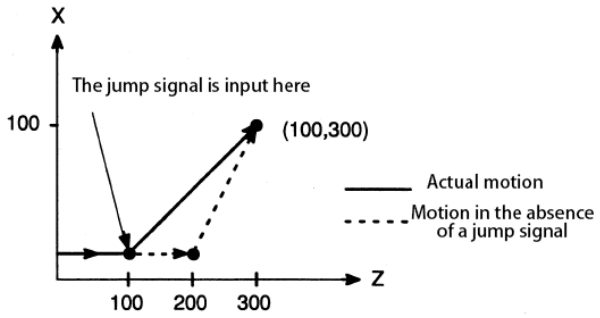


Figure 3-15

Action: When the jump signal changes to "1", the CNC processes as follows:

While the program segment is executing jump code G31, the CNC stores the current absolute coordinate position of each axis. The CNC stops the movement of the G31 code and starts the execution of the next program segment, and the jump signal detects not its rising edge, but its state. Therefore, if the jump signal is "1", it is considered that its jump condition is immediately met**.**

Note: To ensure stop position accuracy, the feed speed of the G31 should be as low as possible.

2.3.18 Brake output

Signal diagnostics

Diagnostic address	Y2.1
Interface pins	CN62.30
Note description	Brake output
remark	The output address is multiplexed with the output address #1106 and M89 I6, please note the choice to use

Relevant parameters

K028.6=1 (0: Brake output 1 : M89 I6 definition / Macro output #1106) Output Y2.1 effect

K	2	8

BIT7	BIT6	BIT5	BIT4	BIT3	BIT2	BIT1	BIT0

BIT7 : (0: Brake output 1 : M89 I6 definition / Macro output #1106) Output Y2.1 effect

- Function description:

After the system is ready to start up, open the brake with a delay of 1 second, and immediately disconnect the brake output when the emergency stop or drive alarm is triggered;

2.3.19 Protective door function

Diagnostic address	X0.0
Interface pins	CN61.1
Note description	Protective door detection signal
remark	The signal is reused with the external processing blowing button

Relevant parameters

K014.2==1 Guard door function (0: invalid 1: valid).

K014.3==1 The protective door detection signal is (0:NORMALLY open 1:NC) is valid

K014.4==0 Spindle when the protective door is open (0: closed 1: not closed).

K 0 20.5==1 (0: external processing blowing button 1: protective door signal) is valid;

K	1	4

BIT7	BIT6	BIT5	BIT4	BIT3	BIT2	BIT1	BIT0

BIT2 : Protective door function (0: invalid 1: valid).

BIT3 : The protective door detection signal is (0:NO 1:NC) active

BIT4 : Spindle when the guard door is open (0: closed 1: not closed).

- **Function description:**

Start the program with the protective door open, system alarm A2.0 [the protective door is not closed, automatic operation is not allowed];

When the door is opened during program operation, the program is suspended, the cooling and processing blowing is closed, and the spindle can choose whether to close or not.

2.3.20 Spindle loose clamping function

Diagnostic address	X0.2	X1.0	X1.1	Y0.2
Interface pins	CN61.03	CN61.09	CN61.10	CN62.03
Note description	External loose/clamp button	The knife is released into place	The tool is clamped in place	Tool loosening/clamping
remark	Normally open contacts	Normally open	Normally open	M16/M17

Relevant parameters

K014.7==1 (0: not used 1: using) Tool clamping device

K013.7==0 External tightening knife input signal selection (0: jog 1: normal).

K013.6==0 System Panel Loose Knife Button (0: Valid 1: Invalid) **Note: Change to 1 when the tool change button on the system panel is not required**

K014.6==0 (0: Check 1: Do not check) Tool clamping/loosening in place signal

K014.5==0 Tool loose clamping control and spindle rotation (0: interlock 1: non-interlock).

K014.0==0 The spindle release signal is (0:normally open 1:normally closed) is valid

K014.1==0 The spindle clamping in place signal is (0:normally open 1:normally closed) is valid

K031.0==0 The clamping signal can be detected when there is no tool after tool clamping (0:Yes 1:No) (this parameter does not detect the tool clamping signal for the tool change electric spindle without tool loading).

K	1	4	BIT7	BIT6	BIT5	BIT4	BIT3	BIT2	BIT1	BIT0

BIT0 : The spindle release signal is (0:NO 1:NC) is valid

BIT1 : The spindle clamping in place signal is (0:NO 1:NC) is valid

BIT5 : Tool loose control and spindle rotation (0: interlock 1: non-interlock).

BIT6 : (0: Check 1: Do not check) Tool clamping/loosening in place signal

BIT7 : (0: not used 1: use) Tool clamping device

K	1	3	BIT7	BIT6	BIT5	BIT4	BIT3	BIT2	BIT1	BIT0

BIT7 : External tightening knife input signal selection (0: jog 1: normal).

BIT6 : System panel loose knife button (0: valid 1: invalid) **Note: Change to 1 when the tool change button on the system panel is not needed**

T	0	7	Tool release completion delay (ms) default 50

T	0	8	Tool clamping completion delay (ms) default 50

T	0	9	Tool release detection delay (ms) defaults to 5000

T	2	0	Tool clamping detection delay (ms) defaults to 5000

T	1	1	9	Tool tightness anomaly detection delay (ms) defaults to 0

- **Function description**

  When the system uses a tool loosening device, it can be controlled manually or with program code M16/M17.

  M16 control output Y0.2 output loose knife, M17 control Y0.2 close clamp knife;

  Control conditions: the spindle is in the stop state and the spindle feedback speed is within the data D126 setting value, and the system is not in tapping mode, otherwise the system will generate an alarm prompt; Manual control can only be carried out when the program is running.

  Manual control of sub-point operation control and normal control by K13.7 switching control mode:

  Jogging mode K13.7=0, when using the external loose/clamping knife button or panel [tool change key] to control loose/clamping knives, press the button once for loose knife, and then press again to clamp the knife, switching back and forth.

  In the normal mode K13.7=1, when using the external loose/clamping knife button or panel [tool change key] to control the loose/clamping knife, press and hold the button for loosening the knife, and release the button for the knife clamping.

  Loose/clamp alarm:

When the spindle rotates or taps, the control tool is released, and the system generates alarm A2.3 (when the spindle rotates, the tool must not be released);

When the spindle control outputs a stop signal, but the spindle speed does not slow down to less than the D126 set value, the control tool is released, and the system generates alarm A10.1 (the spindle speed is too high, the tool cannot be released);

When the spindle tool has been loosened, when the spindle is started or the tapping state is switched, the system generates alarm A2.6 (the spindle tool is loosened and the spindle must not be started);

The system settings check when the tool is loosened/clamped into place signal:

1) The tool is released from the Y0.2 output, and when the loosening signal is not detected within the T09 set time, the system generates alarm A10.5 (spindle tool release detection abnormality).
2) Tool clamping Y0.2 is closed, and when the clamping signal is not detected within the T020 set time, the system generates alarm A10.4 (spindle tool clamping detection abnormality).
3) When the tool release/clamping in place signal is detected at the same time, the system generates alarm A10.7 (spindle loosening knife tightening detection abnormality).
   For the short distance between the elastic in place sensors, and two signals will be sensed at the same time in the process of tightening the knife, the detection time can be extended by setting a slightly larger T119.
4) When the spindle is not jogged, the tool clamping signal is not detected during the spindle rotation, and the system generates alarm A2.4 (when the spindle rotates, the tool clamping signal is invalid alarm).
5) Not in the spindle jog state, the tool clamping signal is not detected to start the spindle, and the system generates alarm A2.5 (the spindle must not be started when the tool clamping signal is invalid).

Note: When the tool loose clamping control is selected and the spindle rotation is not interlocked, the above alarm is invalid

2.3.21 Indexing table function

Relevant signals

Diagnostic address	X0.6	X1.5	X1.6	Y1.6	Y1.7
Interface pins	CN61. 07	CN61.14	CN61.15	CN62.15	CN62.16
Note description	External workbench control	The table is clamped in place	The workbench is released into place	Indexing table clamping output	The indexing table releases the output
Note 1	Normally open	Normally open	Normally open	K4 key M10	K4 key M11
Note 2	Reuse with lubrication	The table clamping loosening in place signal and spindle shift in place signal reuse, pay attention to the selection of use			

	testing	

Relevant parameters:

Bit parameters:

033.3=1 sets the 4th axis to (0: linear axis 1: rotation axis);

034.7=0 Whether it is a four-axis linkage system (0:no 1:Yes)

PLC parameters:

K001.7 == 1 Automatic indexing table (0: forbidden 1: allowed) (when set to 0, 4th axis G0 movement does not automatically release clamping).

K001.6 == 0 Output after the indexing table is in place (0: uninterrupted 1: disconnected) (when set to 1, the output is disconnected after the table is released/clamped into place).

K001.5 == 0 Indexing table in place signal (0: detect 1 : do not detect) (when set to 1, it is not detected after the table is released/clamped into place, but can be set by T41/42 delay).

K001.4 == 0 The indexing table is not clamped (0: forbidden 1: allowed) XYZ axis movement (when set to 1, when not clamped, the XYZ axis can be moved; This parameter is only valid if the automatic indexing table is valid K1.7=1).

K002.7== 1 When the indexing table is clamped in place (0:no 1:Yes), the 4th axis is enabled (when set to 1, the clamping is in place and continuously enabled; This parameter is only valid if the automatic indexing table is valid K1.7=1).

K20.6==0 X0.6 signal (0: lubrication detection 1: external table control) is valid

T040==200 Automatic indexing table clamping delay F20.1

T041==200 The indexing table clamping delay into position (also valid when K1.5 is not detected by bit signal).

T042==200 The indexing table is released into bit delay (also valid when K1.5 is not detected for a bit signal).

T043==10000 indexing table clamping to position delay detection (invalid when K1.5 does not detect bit signal).

T044==10000 The indexing table is released to the position delay detection (invalid when K1.5 does not detect the bit signal).

Function description:

After setting the automatic indexing table is effective (K1.7=1, bit parameter 34.7=0; bit parameter 33.3=1; When the system uses the B axis, it can only use the G0 fast movement command, and when G01 is used, the system alarms; When using the B axis to move, the system sends a signal to release Y1.7, release in place before moving, after the B axis reaches the specified coordinates, the system automatically sends a clamping Y1.6 signal, after clamping in place, the instruction is completed. If the system detects a bit signal, the system will alarm if the system does not detect the corresponding signal within the T043 and T044 time; When no bit signal is detected, T043 and T044 times are not detected.

When there is no detection switch, you can set K001.5 = 1 not to detect the bit signal, but the delay of the signal in place is just as effective, so when the signal is not detected, the time of T041 and T042 can be set to ensure that the clamp is released and tightened in place before completing the instruction.

The system can set whether the XYZ axis can be moved when the table is not clamped (K1.4); At the same time, when the table is clamped, the 4th axis system is moved to alarm; After the table is clamped, the 4th axis can be set to be disconnected, provided that the position parameter 013.5 is set to 1 (the axis enable

signal is controlled by the PLC).

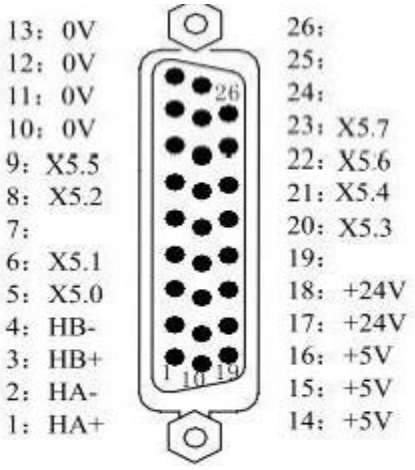
The automatic indexing table loosening clamping control can be controlled by M10/M11 code; You can also manually control the standby state of the system and the stop state of the 4th axis through the manual button [K4 control]; At the same time, when the 4th axis is zeroed, the table is released and the table is clamped after the zeroing is completed.

When the machine tool does not need the indexing table is, Y1.6/Y1.7 can be used as other control functions, the parameter set to the automatic indexing table is invalid (K1.7=0), and the parameter parameter is 34.7=1; The parameter K1.4/K2.7 is invalid.

Simultaneous control only M10/M11 and manual control, moving the 4th axis automatic release clamping and 4th axis zero release clamping is ineffective.

2.4 Handwheel connection setup instructions

2.4.1 Handwheel Interface Definition



CN31 handwheel connector (26-pin pin).

address	Signal	illustrate
1、2	HA + 、HA -	Handwheel A phase signal
3、4	HB + 、HB -	Handwheel Phase B signal
5	X5.0	X-handwheel axle selection
6	X5.1	Y-handwheel axle selection
8	X5.2	Z-handwheel axle selection
20	X5.3	4-axis handwheel axle selection
21	X5.4	5-axis handwheel axle selection
9	X5.5	Increment X1
22	X5.6	Increment X10
23	X5.7	Increment X100
17、18	+24V	DC power supply 24V
14、15、16	+5V	DC power supply 5V
10-13	0V	DC power supply 0V

Control parameters

Status parameter

013.0==0 The coordinates increase when the handwheel (0: forward 1: counterclockwise) rotates

013.7==1 The handwheel wheel rotation displacement (0:No 1:Yes) runs in full

Note: Position 13.6 [Handwheel test cut (0:no 1:Yes) can run in direction].

0	1	3

BIT7	****	****	BIT4	BIT3	BIT2	BIT1	BIT0

BIT0 : The coordinate increases when the handwheel (0: forward 1: counterclockwise) rotates

BIT7 : Handwheel wheel rotation displacement (0:no 1:Yes) runs in full

Data parameters

3	3	3	Maximum clamping speed for incomplete operation of the handwheel (default: 2000).

PLC parameters

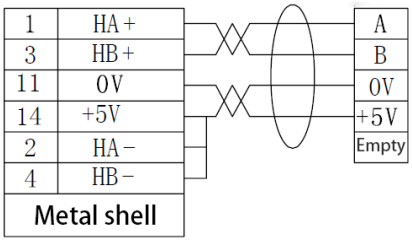
K3.4==0 External handwheel axle selection (0: valid 1: invalid).

K	0	3	****	****	****	BIT4	****	****	****	****

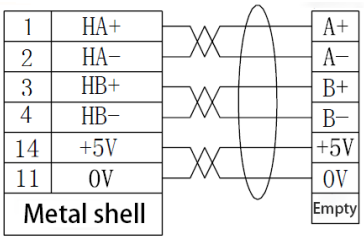
BIT4 : External Hand Axle Selection (0: valid 1: invalid).

External signal connection of the machine

(CN31) handwheel (CN31) handwheel



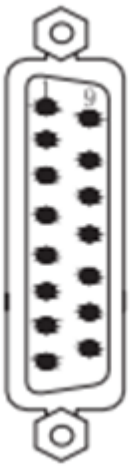
Single-ended method



Differential connection

2.5 The spindle is connected to the spindle encoder

2.5.1 Encoder interface definition



Foot size	name	illustrate
11、14、15	GND	Power supply 0V
3	#PCS	Encoder phase C negative pulse
4	PCS	Encoder phase C forward pulse
5	#PBS	Encoder phase B negative pulse
6	PBS	Encoder phase B forward pulse
7	#PAS	Encoder A phase negative pulse
8	PAS	Encoder A phase forward pulse
12	TH5IO5V	Power supply 5V
13	TH5IO5V	Power supply 5V

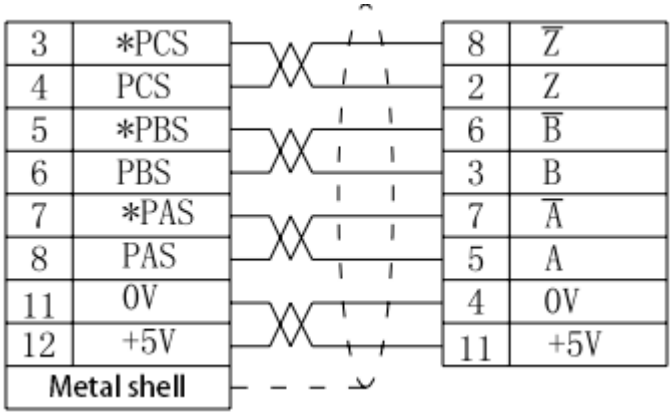
CN21encoderinterface(15-pin pin**).**

2.5.2 The spindle is connected to the encoder

The connection of the milling machine series to the spindle encoder is shown in the figure below, using twisted pair when connected. (Take the light of Changchun.)

ZLF-12-102.4BM-C05D encoder for example):

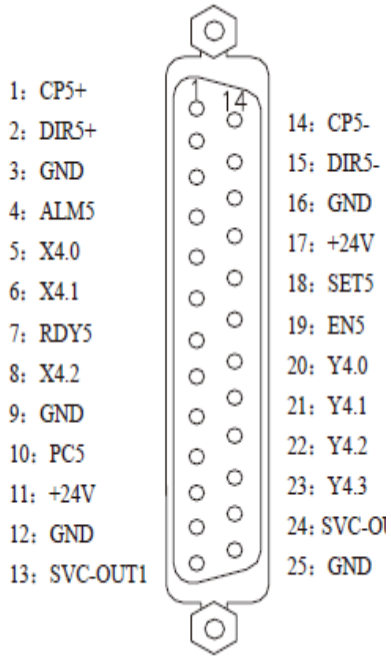
Milling machine series Changchun Yiguang 1024 encoder



Data parameters

2	0	1	Number of spindle encoder lines (default: 1024).
2	2	0	Encoder to spindle gear ratio parameters: number of gears on the spindle side [1st gear] (default value: 1).
2	2	1	Encoder to spindle gear ratio parameters: number of spindle side gears [2nd gear] (default value: 1).
2	2	2	Encoder to spindle gear ratio parameters: number of spindle side gears [3rd gear] (default value: 1).
2	2	3	Encoder to spindle gear ratio parameters: the number of gears on the encoder side [1st gear] (default value: 1).
2	2	4	Encoder to spindle gear ratio parameters: the number of gears on the encoder side [2nd gear] (default value: 1).
2	2	5	Encoder to spindle gear ratio parameter: the number of gears on the encoder side [3rd gear] (default value: 1).

2.5.3 Spindle Interface Definition



CN15 5th axis spindle interface

Foot size	address	definition
3、9、6	0V	Power supply 0V
11、17	+24V	Power supply +24V
1	CP5+	The spindle pulse commands a positive signal
2	DIR5+	Positive signal in the direction of the spindle pulse
4	ALM5	Spindle drive alarm
5	X4.0	6^{TH} return to zero deceleration signal
6	X4. 1	Spindle speed arrival / 2nd spindle alarm
7	RDY5	The spindle is ready for signaling
8	X4.2	The spindle is directed to the signal in place
9	GND	Power supply 0V terminal
10	PC5	Spindle zero signal
11	+24V	The power supply is +24V
12	GND	The spindle analog voltage is 0V
13	SVC-OUT1	Spindle analog voltage output 1
14	CP5 -	The spindle pulse commands a negative signal
15	DIRD -	Negative signal in the direction of the spindle pulse
19	EN5	Enable signal
20	Y4.0	Spindle position control output
21	Y4.1	Pulse spindle enable/spindle forward rotation
22	Y4.2	Spindle directional output
23	Y4.3	The spindle is reversed
24	SVC-OUT2	Spindle analog voltage output 2
25	GND	The spindle analog voltage is 0V

Note 1: X4.0-X4.2 detection +24V on-off is effective; The spindle alarm signal detects that 0V on and off is effective.

Note 2: Y4.1 is a signal that will be output by both forward and reverse and tapping modes of the pulse spindle, in order to facilitate the connection of the full pulse control spindle enabling effect.

Note 3: Y4.1 is the composite output point, which can be enabled for the pulse spindle **or the spindle forward output by K 17.2 selection function, and enabled for the pulse spindle when K17.2=0; K17.2=1 forward rotation of the spindle (the function is the same as Y0.3 M03 output, M05 off).**

Note 4: Y 4.3 is the main axis reverse (the function is the same as Y0.4 M04 output, M05 off).

Note 5: The PC5 zero signal is valid with 0 **V, and no resistor is required for wiring (the other 4 feed axes PC signal is valid with 24V);**

Note: Spindle port assignment with Axis 5 or Axis 6 or Axis 7 hardware;

1: The normal version hardware has 5 axes, and the spindle port can choose the 4th axis or the 5th axis;
2: The TI hardware version has 6 axes, the spindle port can be selected as a 4th axis or 5th axis or 6th axis;
3: The bus version hardware has 6 feed axes and 1 auxiliary axis, and the auxiliary axis is the 7th axis, which only has the function of pulse spindle and tapping;

The port selection has parameters to determine:
Number parameter 326 Spindle pulse port selection (4:4th 5 or other: 5th 6:6th 7:7th) (default 5th axis).
Bit parameter 5 0 Rigid tapping selects axis (0: 5th axis 1: 4th axis) (default 5th axis).
Bit parameter 5 2 Rigid tapping selects the 6th axis (0: invalid 1: effective).
Bit parameter 5 7 Rigid tapping selects the 7th axis (0: invalid 1: valid).
Other selections have no effect when selecting axis 7, and axis 4/5 are invalid when axis 6 is selected

2.5.4 Spindle Control

Relevant signals (defined by standard PLC programs).

Signal type	symbol	Signal interface	address	Signal function	remark
Input signal	nALM_M	CN15.04		Spindle abnormal alarm input	This signal has a valid 0V input
Output signal	M03	CN62.04	Y0.3	The spindle rotates counterclockwise (forward rotation).	Y4.1 function selects forward rotation
	M04	CN62.05	Y0.4	The spindle rotates clockwise (reverses).	Y4.3 Functional reversal
	M05	CN62.06	Y0.5	The spindle stops	Address multiplexing, please note the option to use
	SPZD	CN62.08	Y0.7	Spindle brake	Address multiplexing, please note the option to use
	GND	CN15.12		The spindle analog voltage is 0V	
	SVC_OUT1	CN15.13		Spindle analog voltage 10V output	
Instruction format	M03			The spindle rotates counterclockwise (forward rotation).	
	M04			The spindle rotates clockwise (reverses).	
	M05			The spindle stops	

Note: Spindle stop output address Y0.5 and macro output #1100, M89 I0 output address multiplexing

K28.0 selection; spindle brake output address Y0.7 and macro output #1102, M89 I2 output address multiplexing K28.2 selection;

Control parameters

Status parameter

001.4==0 Spindle speed (0: analog voltage control 1: switching control).

009.7==1 Spindle alarm signal (0:high 1:low) level alarm (disconnected from 0V).

0	0	1	****	****	****	BIT4	****	****	****	****

BIT4 : Spindle speed (0: analog voltage control 1: switching control).

0	0	9	BIT7	****	****	****	****	****	****	****

BIT7 : Spindle alarm signal (0:high 1:low) level alarm (on and off with 0V).

Data parameters

2	0	6	Spindle analog voltage output voltage offset offset compensation value (default value: 0).
2	1	0	Corresponds to the maximum speed of the first gear (M41) of the spindle [Set the maximum speed of the spindle] (default: 6000).
2	0	1	Spindle encoder line speed (default: 1024).
2	0	0	Maximum spindle speed (default: 6000).
2	0	8	Rotation speed when the spindle is jogged (default: 40).
2	2	0	Encoder to spindle gear ratio: the number of spindle end gear teeth (default: 1).
2	2	1	Encoder to spindle gear ratio: the number of gear teeth at the encoder end (default: 1).
3	7	4	The spindle speed value is set at the system power-up

- **Spindle jog function:**

T	1	2	Spindle jog time (default: 3000) unit: milliseconds

In the spindle analog voltage control state, turn on the [spindle jog] function, and then manually select the spindle forward/reverse; The spindle rotates forward or reverses according to the speed set by parameter 208, and stops after entering the time set by PLC parameter T12.

- **Spindle brake function:**

T	1	0	Spindle brake delay output time (ms) (default: 0).

T	1	1	Spindle brake output time (ms) (default: 50).

In the analog voltage control state of the spindle, the spindle stops after forward/reverse; The spindle passes through the time delay set by PLC parameter T10

The rear output spindle brakes and is disconnected after the time set by PLC parameter T11.

Warning: The spindle cannot be started when the tool changer is not at the origin position.

Note: If you choose the pulse control spindle speed mode, you can select the 4th axis **or 5th axis shaft port pulse control spindle, pay attention to the selection;**

Number parameter 174 The feed shaft does the pulse spindle stop minimum speed (default 1).

Number parameter 047 The maximum speed of the second pulse spindle (default 2000).

Bit parameter 007. 3 When the feed shaft does the pulse spindle stop (0:no 1:yes), it stops in the pre-start position

Bit parameter 007. 4 Whether the 5th axis is set to the pulse spindle (0:no 1:Yes)

Bit parameter 007. 5 The 4th axis is set as the pulse spindle (0:No 1:Yes).

Bit parameter 022. 4 The 4th axis is the analog spindle selection (0: 1st spindle 1: 2nd spindle).

Bit parameter 022. 5 The 5th axis is the analog spindle selection (0: 1st spindle 1: 2nd spindle).

If the pulse spindle parameters 7.4=1 and 7.5=1 are selected for both the 4th and 5th axes, select 1st When the spindle position parameters 2 2.4=0 and 2 2.5=0, the spindle rotation command M3/M4 is started , the two axes send the same pulse to control the spindle; So pay attention to the choice of use.

5th axis 2nd spindle function standard layout ladder diagram does not retain the write function;

The 4th axis and the second spindle function are written in the second spindle control, and can be controlled by M63/M64/M65 after the parameters are opened;

Parameter 7.3 (**when the feed shaft stops as a pulse spindle (0:no 1:Yes) stops at the pre-start position) parameter role:**

When 7.3=0 is not selected to stop at the start bit, the stop position of the **pulse spindle is random, but the stop position corresponds to the coordinate position displayed by the system;**

Select Stop at the start time 7.3=1, the pulse spindle stop position will stop the coordinate position before starting; The minimum speed of the stop has several parameters 174 control.

The control pulse spindle gear ratio will be changed to feed shaft gear ratio control;

The gear ratio of the pulse spindle in the fifth axis is 4/9;

The gear ratio of the 4th shaft pulse spindle is 3/8;

The gear ratio calculation is consistent with the feed shaft algorithm, for example, the number of encoder lines of the spindle motor is 2500, **and the rotation shaft goes in one turn 360,**

Numerator/denominator = [2500*4]/[360*1000]=10/360;

Under the normal gear ratio of the pulse spindle 1/36, the maximum pulse output frequency is 6 000r/min, if the spindle needs to be greater than 6 000r/min, the system gear can be magnified by 2 times than the denominator, and the driven molecule can be magnified by 2 times, so that the output can be 12000r/min, and so on.

Note: A 6th-axis analog spindle is added to a 6-axis hardware interface or bus version system

Parameter 7.0 Axis 6 sets the analog spindle (0: invalid 1: valid).

Parameter 7.1 6th axis analog spindle (0: 1st spindle 1: 2nd spindle).

Parameter 8.5 6th axis movement direction (0: not reverse 1: negation) can control the forward and reverse rotation of the spindle and the tapping direction

Number parameter 420 6th axis molecule

Number parameter 421 6th axis denominator

Number parameter 442 Linear acceleration and deceleration time constant of linear type in front acceleration and deceleration of the 6th axis (simulated spindle acceleration and deceleration time constant).

Number parameter 443 S-type time constant for acceleration and deceleration before rapid movement of axis 6

Note: A 7th-axis analog spindle has been added to the bus version system

Parameter 2.0 Whether the 7th axis is set to the pulse spindle (0:no 1:Yes).

Number parameter 96 7th axis molecule

Number parameter 97 7th axis denominator

Number parameter 98 Linear acceleration and deceleration time constant of the 7th axis front acceleration and deceleration type (simulated spindle acceleration and deceleration time constant).

Note: The bus system usesM3 bus spindle parameters

bit parameter 51.4 [M3 communication last address as bus spindle function (0: invalid 1: valid)];

Parameter 5 1.5 [M3 bus spindle speed feedback (0: external encoder 1: M3 bus feedback)];

2.5.5 Spindle speed switching control

Relevant signals (defined by standard PLC program, no switching control in ladder 2).

S01~S04: spindle speed switching control signal, standard PLC program definition of S01~S04 signal interface for multiplexing interface, S01~S04 Shared interface with M41~M44.

Control logic (standard PLC program definition).

When the CNC is powered up, the S1 ~ S4 output is invalid. Execute any code in S01, S02, S03, S04, the corresponding S signal output is valid and maintained, while canceling other S The output of the signal. When executing the S05—S07 code, cancel the output of S1 ~ S4, S1 ~ S4 Only one output is valid at a time.

Signal diagnostics

Signal	S1	S2	S3	S4
Diagnostic address	Y1.0	Y1.1	Y1.2	Y1.3
Interface pins	CN62.09	CN62.10	CN62.11	CN62.12
Note description	Spindle 1st gear	Spindle 2nd gear	Spindle 3 gears	Spindle 4 gears
remark	The spindle gear output address is multiplexed with the macro output #1102--#1105, M89 I2-I5 output address, please pay attention to choose to use			

Control parameters

Status parameter

001.4==1 Spindle speed (0: analog voltage control 1: switching control).

0	0	1	****	****	****	SPTY	****	****	****	****

BIT4 : Spindle speed (0: analog voltage control 1: switching control).

PLC parameters

T	0	6	Code execution time (default: 1).

PLC parameters

K015.4==0 Delayed disconnect output after spindle shift (0: invalid 1: valid).

K028.2==0 (0: spindle 1 stop 1: M89 I2 custom/macro output #1103) Output Y1.0 is valid

K028.3==0 (0: spindle 2 gears 1 : M89 I3 custom/macro output #1104) Output Y1.1 is valid

K028.4==0 (0: spindle 3 gears 1: M89 I4 custom/macro output #1105) Output Y1.2 is valid

K028.5==0 (0: spindle 4 gears 1: M89 I5 custom/macro output #1106) Output Y1.3 is valid

K	1	5	****	****	BIT5	BIT4	BIT3	BIT2	BIT1	BIT0

BIT4 : Delayed disconnect output after spindle shift (0: invalid 1: valid).

K	2	8	BIT7	BIT6	BIT5	BIT4	BIT3	BIT2	BIT1	BIT0

BIT0 : (0: Spindle Stop 1: M89 I0 Custom/Macro Output #1100) Output Y0.5 is valid

BIT1 : (0: Spindle brake 1: M89 I1 custom/macro output #1101) Output Y0.6 valid

BIT2 : (0: spindle 1 stop 1: M89 I2 custom/macro output #1102) Output Y0.7 valid

BIT3 : (0: spindle 2 gears 1: M89 I3 custom/macro output #1103) Output Y1.0 valid

BIT4 : (0: spindle 3 gears 1 : M89 I4 custom/macro output #1104) Output Y1.1 is valid

BIT5 : (0: spindle 4 gears 1: M89 I5 custom/macro output #1105) Output Y1.2 valid

BIT6 : (0: Brake output 1: M89 I6 custom/macro output #1106) Output Y1.3 is valid

BIT7 : (0: 2nd spindle stop output 1: M89 I7 custom/macro output #1107) Output Y2.7 valid

Note: When K15.4=1 [Delay to disconnect output after spindle shift is completed (0: invalid 1: valid)], the delay after spindle shifting is completed is 0 Disconnect the gear output for 3 seconds.

2.5.6 Automatic spindle shift control

Relevant signals (defined by standard PLC programs).

M41 ~ M44: Spindle automatic shift output signal, when spindle analog control (0 ~ 10V analog voltage output) is selected, it can support 4 gears of spindle automatic shift control.

M41I, M42I: Spindle automatic shift 1st and 2nd gear shift in place signal, can support 2 gear shift in place detection function.

Signal diagnostics

Signal	M41I	M42I	M41	M42	M43	M44
Diagnostic address	X1.5	X1.6	Y1.0	Y1.1	Y1.2	Y1.3
Interface pins	CN61.14	CN61.15	CN62.09	CN62.10	CN62.11	CN62.12
Note description	1 stop in place	2 stops in place signal	Spindle 1st gear	Spindle 2nd gear	Spindle 3 gears	Spindle 4 gears
remark	The in-place signal is multiplexed with the workbench clamp-in-place signal		The spindle gear output address is multiplexed with the macro output #1102--#1105, M89 I2-I5 output address, please pay attention to choose to use			

Control parameters

Status parameter

001.4==0 Spindle speed (0: analog voltage control 1: switching control).

0	0	1	****	****	****	SPTY	****	****	****	****

BIT4 : Spindle speed (0: analog voltage control 1: switching control).

PLC parameters

K015.0==1 Spindle automatic shift function (0: invalid 1: valid).

K015.1==0 (0: Do not check 1: Check) Shift in place signal

K015.2==1 The automatic shift 1 and 2nd gears in place signal is (0:normally open 1:normally closed) is valid

K015.3==0 Spindle gear (0:No 1:Yes) Power-down memory

K015.4==0 Delayed disconnect output after spindle shift (0: invalid 1: valid).

K015.5==0 spindle forward rotation signal when spindle automatic gear change (0:no 1:Yes).

K028.3==0 (0: spindle 1 stop 1: M89 I3 custom/macro output #1103) Output Y1.0 is valid

K028.4==0 (0: spindle 2 gear 1 : M89 I4 custom/macro output #1104) Output Y1.1 is valid

K028.5==0 (0: spindle 3 gears 1: M89 I5 custom/macro output #1105) Output Y1.2 is valid

K028.6==0 (0: spindle 4 gears 1: M89 I6 custom/macro output #1106) Output Y1.3 is valid

K	1	5	****	****	BIT5	BIT4	BIT3	BIT2	BIT1	BIT0

BIT0 : Spindle automatic shift function (0: invalid 1: valid).

BIT1 : (0: Do not check 1: Check) Shift in place signal

BIT2 : Automatic gear shift 1 and 2nd gear in place signal is (0: normally open 1: normally closed) is valid

BIT3 : Spindle gear (0:No 1:Yes) Power-down memory

BIT4 : Delayed disconnect output after spindle shift (0: invalid 1: valid).

BIT5 : Spindle forward rotation signal when spindle automatic gear shifting (0:No 1:Yes).

K	2	8	BIT7	BIT6	BIT5	BIT4	BIT3	BIT2	BIT1	BIT0

BIT0 : (0: Spindle Stop 1: M89 I0 Custom/Macro Output #1100) Output Y0.5 is valid

BIT1 : (0: Spindle clamping 1: M89 I1 custom/macro output #1101) Output Y0.6 is valid

BIT2 : (0: Spindle brake 1: M89 I2 Custom/Macro Output #1102) Output Y0.7 valid

BIT3 : (0: spindle 1 stop 1: M89 I3 custom/macro output #1103) Output Y1.0 valid

BIT4 : (0: spindle 2 gears 1 : M89 I4 custom/macro output #1104) Output Y1.1 is valid

BIT5 : (0: spindle 3 gears 1: M89 I5 custom/macro output #1105) Output Y1.2 is valid

BIT6 : (0: spindle 4 gears 1: M89 I6 custom/macro output #1106) Output Y1.3 valid

BIT7 : (0: Brake output 1: M89 I7 custom/macro output #1107) Output Y2.7 is valid

T	0	0	Spindle shift time 1 (default: 1000) The timing of the spindle shift off the original gear time
T	0	1	Spindle shift time 2 (default: 1000) The delay from the output of the new gear of the spindle to the end
T	4	5	Spindle automatic shift timing ms (default: 10000) time after the time arrives without a detection signal alarm

Data parameters

2	1	0	Corresponds to the maximum speed of the first gear (M41) of the spindle (default value: 6000).
2	1	1	Corresponds to the maximum speed of the second gear (M42) of the spindle (default value: 6000).
2	1	2	Corresponds to the maximum speed (default: 6000) in the third gear (M43) of the spindle
2	1	3	Corresponds to the maximum speed of the spindle in the fourth gear (M44) (default: 6000).
2	1	4	The output voltage (mV) when the spindle is shifting gears (default: 100).

- **Function description:**

1) When the spindle speed is selected as an analog voltage control mode (the Bit4 bit of the status parameter NO.001 is set to 0), and the status parameter K0015.0 is set to 1, the spindle automatic gear shift function is effective; when the spindle automatic shift function is invalid, the CNC will alarm when M41~M44 is executed. M41, M42, M43, M44 are valid only one at a time.

2) The spindle automatic gear shift function is used to control the automatic switching of the spindle mechanical gear, when the CNC executes the S□□□□ code, according to the current M4n control gear corresponding parameters (M41~M44 corresponding to the data parameters NO.210~NO.213) to calculate the analog voltage output to the spindle servo or inverter, the actual speed of the control spindle is consistent with the speed of the S code.

3) When the parameter K0015.3 is 0, the spindle gear is not remembered, and the M41 output is re-powered; when the parameter K0015.3 is 1, the spindle gear is remembered, and the state when the power is turned off will be maintained when the power is re-powered.

4) Execute any code in M41, M42, M43, M44, and output the analog voltage to the spindle servo or inverter according to the value set by data parameter NO.214 (unit: millivolts);

5) After delaying the PLC parameter T000 (shift time 1), turn off the original gear output signal and output a new gear shift signal at the same time;

6) When the gear shift is 1 or 2 gears, and the parameter K0015.1 [(0: do not check 1 : check) shift in place signal] is 1, then turn 7), otherwise turn 8);

7) Check the input signal M41I and M42I in 1 or 2 gears, and turn 8 if the gear is shifted into place); If the gear shift is not in place, the CNC has been waiting for the shift in place signal to pass the time T45 set time and the signal is not detected, then alarm;

8) Delay PLC parameter T001 (shift time 2), according to the current gear according to the data parameter NO.210~NO.213 (corresponding to 1~4 gears) set value output spindle analog voltage, shift end.

9) M4 □ is performed when K15.5=1 [spindle forward rotation signal when spindle automatically shifts (0:no 1:yes)]. When the output gear is output at the same time, the spindle forward rotation is output, and the spindle forward rotation is disconnected after the gear shift is completed

10) When K15.4=1 [Delay disconnect output after spindle shift is completed (0:invalid 1: valid)], the gear output is disconnected by 0.8 seconds after the spindle shift is completed.

2.5.7 2nd spindle function

Relevant signals (defined by standard PLC programs).

Signal type	symbol	Signal interface	address	Signal function	remark
Input signal	ALM	CN15.06	X4.1	2nd spindle alarm signal	Function and speed reach signal multiplexing
Output signal	M63	CN62.36	Y2.7	The second spindle rotates counterclockwise (forward rotation).	
	M64	CN62.37	Y2.6	The second spindle rotates clockwise (reverses).	

	GND	CN15.25		The second spindle analog voltage 0V terminal	
	SVC_OUT1	CN15.24		2nd spindle analog voltage 10V output	
Instruction format	M63			The second spindle rotates counterclockwise (forward rotation).	
	M64			The second spindle rotates clockwise (reverses).	
	M65			The spindle stops	

Note 1: In analog voltage control mode, the second spindle is active;

Note 2: The second spindle magnification is fixed 100% and cannot be adjusted.

Control parameters

Status parameter

001.4==0 Spindle speed (0: analog voltage control 1: switching control).

045.2==1 Whether to activate the multi-spindle control function (0:No 1:Yes).

0	0	1	****	****	****	BIT4	****	****	****	****

BIT4 : Spindle speed (0: analog voltage control 1: switching control).

0	4	5	****	****	****	****	****	BIT2	****	****

BIT2 : Whether to activate the multi-spindle control function (0:no 1:Yes).

Data parameters

0	4	8	The second spindle corresponds to a maximum speed of 1 0V (default: 6000).

Note: At the same time, it is controlled by the maximum speed of the spindle No. 200

2	0	0	Maximum spindle speed (default: 6000).
3	7	4	The spindle speed value is set at the system power-up
0	4	7	The maximum speed of the second pulse spindle

PLC parameters

K17.7==1 Multi-spindle function (0: invalid 1: valid).

K17. 3==0 The second spindle alarm signal is (0:NO 1:NC) is valid

K	1	7	BIT7	BIT6	BIT5	BIT4	BIT3	BIT2	BIT1	BIT0

BIT7 : Multi-spindle function (0: invalid 1: valid).

Note: The 4th axis is used as the second spindle;

Number parameter 174 The feed shaft does the pulse spindle stop minimum speed (default 1).

Number parameter 047 The maximum speed of the second pulse spindle (default 2000).

Bit parameter 007. 3 When the feed shaft does the pulse spindle stop (0:no 1:yes), it stops in the pre-start position

Bit parameter 007. 5 The 4th axis is set as the pulse spindle (0:No 1:Yes).

Bit parameter 022. 4 The 4th axis is the analog spindle selection (0: 1st spindle 1: 2nd spindle).

The 4th axis 2nd spindle function standard layout ladder diagram is written in the 2nd spindle control, and the parameter is opened after the parameter 7.5=1/22.4=1, which is available M63/M64/M65 control;

Parameter 7.3 (when the feed shaft stops as a pulse spindle (0:no 1:Yes) stops at the pre-start position) parameter role:

When 7.3=0 is not selected to stop at the start bit, the stop position of the **pulse spindle is random, but the stop position corresponds to the coordinate position displayed by the system;**

Select Stop at the start time 7.3=1, the pulse spindle stop position will stop the coordinate position before starting; The minimum speed of the stop has several parameters 174 control.

The control pulse spindle gear ratio will be changed to feed shaft gear ratio control;

The gear ratio of the pulse spindle in the fifth axis is 4/9;

The gear ratio of the 4th shaft pulse spindle is 3/8;

The gear ratio calculation is consistent with the feed shaft algorithm, for example, the number of encoder lines of the spindle motor is 2500, and the rotation shaft goes in one turn 360,

Numerator/denominator = [2500*4]/[360*1000]=10/360;

Under the normal gear ratio of the pulse spindle 1/36, the maximum pulse output frequency is 6 000r/min, if the spindle needs to be greater than 6 000r/min, the system gear can be magnified by 2 times than the denominator, and the driven molecule can be magnified by 2 times, so that the output can be 12000r/min, and so on.

2.5.9 Multi-spindle function description

Relevant signals (defined by standard PLC programs).

Signal type	symbol	Signal interface	address	Signal function	remark
Output signal	M03	CN62.04	Y0.3	The first spindle rotates counterclockwise (forward rotation).	Y4.1 function selects forward rotation
	M04	CN62.05	Y0.4	The first spindle rotates clockwise (reverses).	Y4.3 Functional reversal
	M05	CN62.06	Y0.5	The first spindle stops	
	GND	CN15.12		The first spindle analog voltage 0V terminal	

	SVC_OUT1	CN15.13		The first spindle analog voltage 10V output	
	M63	CN62.35	Y2.7	The second spindle rotates counterclockwise (forward rotation).	
	M64	CN62.36	Y2.6	The second spindle rotates clockwise (reverses).	
	M65			The second spindle stops	
	GND	CN15.25		The second spindle analog voltage 0V terminal	
	SVC_OUT2	CN15.24		2nd spindle analog voltage 10V output	

Related instructions

M03 The first spindle rotates
M04 The first spindle is reversed
M05 Spindle 1 stops
M63 2nd spindle forward rotation
M64 2nd spindle reversed
M65 2nd spindle stops

- **Feature description**

1) After the multi-spindle control function is set effectively, the power-on defaults to the first spindle control;
2) Execute M03, M04, M05, the corresponding signal output of the first spindle, at this time will be set to the first spindle control, after which the executed S command will change the No. . 1 spindle speed. Similarly, the corresponding signal output of the second spindle is performed for M63, M64, M65, and the second spindle will be converted to the second spindle control and then executed The S command will change the speed of the second spindle.
3) Spindle switching is ineffective when rigid tapping.
4) The panel spindle manual forward, reverse, stop, spindle magnification buttons are only valid for the first spindle. When currently controlling the second spindle, the panel spindle manual control button can still control the first spindle.
5) The spindle magnification of the second spindle is fixed at 100%.

2.5.10 Spindle speed/position (tapping) mode switching

Relevant signals (defined by standard PLC programs).

Signal type	symbol	Signal interface	address	Signal function	remark
Output signal	M29	CN15.20	Y4.0	The first spindle switches the position mode	Tapping mode
	M28			Eliminate rigid tapping	
Input signal					

Related instructions

M29 Turn on tapping mode

M28 Cancels tapping mode

Relevant parameters

Note: With 6-axis hardware interface or bus version system new 6-axis tapping option

Tap to select the axis

Parameter 5.0 [tapping selection (0: 5th axis 1: 4th axis)].

Parameter 5.2 [tapping selection 6th axis (0: invalid 1: valid)].

Parameter 5.7 [tapping selection 7th axis (0: invalid 1: valid)].

When selecting the 6th axis is valid, selecting the 4/5 axis is invalid;

When selecting the 7th axis is valid, selecting the 4/5/6 axis is invalid;

Status parameter

001.4==0 Spindle speed (0: analog voltage control 1: switching control).

005.0==0 Rigid tapping selection 4th axis (0:no 1:Yes) [When 0 is selected, pulse tapping is sent with the pulse hole of the 5th axis; When 1 is selected, pulse is sent out at the 4th axis pulse port].

031.0==1 Pitch F setting for rigid tapping (0:No 1:Yes) [When 0 is selected, the tapping section is programmed for speed F.] = spindle speed * tap pitch; When 1 is selected, the tapping section is programmed with speed F value = tap pitch].

058.0==1 The spindle control mode is (0:Follow 1:Servo) [follow is flexible tapping, servo is rigid tapping].

058. 1==0 Whether the tapping becomes a high-speed deep hole tapping cycle (0:No 1:Yes) [When the high-speed deep-hole tapping cycle is not selected, when editing G84Q_, the Z-axis retract retreats to point R.] , and then tap back and forth; When the high-speed tapping cycle is selected, when editing the G 84Q_, the Z-axis retract distance is set to the parameter value (number parameter 3 3No. 9[The amount of backoff during the deep hole tapping cycle (default: 2)]) is then reciprocated and tapped.

058.6==0 Whether the magnification is valid when rigid tapping and retracting (0:no 1:Yes) [When the retract magnification is invalid, the tapping and retracting speeds are the same; When the retract magnification is selected, the tapping and retracting speed is subject to the number parameter 3 38 (the magnification value of rigid tapping and retracting (retract speed = F* value of the approach speed)), and the number parameter 3 38 = 1 , the retract speed and the approach speed are the same, and when the number parameter 338=1.2, the retract speed is 1.2 times the approach speed].

0	0	1	****	****	****	BIT4	****	****	****	****

BIT4 : Spindle speed (0: analog voltage control 1: switching control).

0	0	5	BIT7	****	****	****	BIT3	BIT2	****	BIT0

BIT0 : Rigid tapping selects 4th axis (0:No 1:Yes)

0	3	1	BIT7	****	****	****	BIT3	BIT2	****	BIT0

BIT0 : Pitch F setting for rigid tapping (0:No 1:Yes)

0	5	8	****	****	****	****	****	****	****	BIT0

BIT0 : The spindle control mode is (0:follow 1:servo) [follow for flexible tapping, servo for rigid tapping].

BIT1 : Whether the tapping becomes a high-speed deep hole tapping cycle (0:No 1:Yes).

BIT6 : Whether the magnification is valid when rigid tapping and retracting (0:no 1:Yes)

Data parameters

2	3	0	Tapping spindle command multiplication coefficient CMR (tapping pulse molecule) (1st gear gear).
2	3	1	Tapping spindle command multiplication factor CMR (tapping pulse molecule) (2nd gear gear).
2	3	2	The tapping spindle commands the multiplication factor CMR (tapping pulse molecule) (3rd gear gear).

2	3	3	The spindle commands the frequency division factor CMD (tapping pulse denominator) (1st gear gear).
2	3	4	The spindle commands the frequency division factor CMD (tapping pulse denominator) (2nd gear gear).
2	3	5	The spindle commands the frequency division factor CMD (tapping pulse denominator) (3rd gear).

Note: 2500 line number motor 230=10;233=1;1024 line number motor 230=512;**233=125;**

Note: 230=**10**; **233=1 when the system defaults to spindle rotation of 10,000 pulses, other gear ratios can be set in the drive;**

PLC parameters

T	2	5	M29 completion delay Unit: milliseconds; Default 50

T	2	7	M28 completion delay Unit: milliseconds; 50

- **Function description:**

When executing the tapping mode M29 code, Y4.0 (position mode switching) is output, and the system delay T025 sets the time to complete the M29 code.

Execute the M28 code to cancel the tapping mode to disconnect the Y4.0 output, and the system delay T027 set time to complete the M28 code.

Note: When the spindle brake output is output, you need to wait for the spindle brake output to be completed before exporting the position mode.

- **To cancel the rigid tapping mode:**

  In the automatic running state, the tapping mode can be canceled by running M28 code, program running end, spindle positioning code M19, spindle forward and reverse M3/M4;

  In manual untapping mode, turn on spindle jogging, spindle positioning, spindle forward/reverse/stop buttons.

- **Tapping programming mode:**

  M29 S_;//S_ tapping speed (M 29 command can be ignored).

  G84 Z-10 F1.5 R5 ;//right-handed tapping; G74 left-handed tapping

  G80;//End of tapping code

2.5.11 Spindle orientation function

Signal type	symbol	Signal interface	address	Signal function	remark
Output signal	M19	CN15.22	Y4.2	Spindle directional output	
	M18			Unorient the spindle	
Input signal		CN15.08	X4.2	The spindle directs the completion signal	Valid for +24V on

Related instructions

M19 spindle positioning output

M18 Unposition the spindle

Relevant parameters

Status parameter

001.4==0 Spindle speed (0: analog voltage control 1: switching control).

0	0	1	****	****	****	BIT4	****	****	****	****

BIT4 : Spindle speed (0: switching control 1: analog voltage control).

PLC parameters

K021.1==0 The spindle directional completion signal is (0:NO 1:NC) is valid

K	2	1	BIT7	BIT6	BIT5	BIT4	BIT3	BIT2	BIT1	BIT0

BIT1 : The spindle directional completion signal is (0:NO 1:NC) active

T	0	4	Spindle orientation completion delay unit: milliseconds; Default 50
T	1	8	Spindle directional delay detection unit: milliseconds; Default 5000
T	2	3	Spindle directional completion signal delay detection Unit: milliseconds; Default 2000
T	1	1	8	Positioning completion signal disconnection delay Unit: milliseconds; Default 300

- **Function description:**

  When the system does not send a spindle orientation signal, the system sends a spindle orientation signal when the system does not press the [spindle directional key] or execute the spindle positioning M19 or perform fine boring G76/back boring cycle G87 Y4.2, wait for the spindle orientation completion signal X4.2 before the code can be completed. When the directional signal Y4.2 is sent, the directional completion signal is not received at the T018 set time, and the system alarm A13.1 (spindle orientation in place detection abnormality); When the system receives the spindle orientation completion signal level setting error, the spindle sends a directional signal system to generate alarm A12.5 (spindle directional timing detection abnormality); When the spindle tool is released, the spindle orientation cannot be performed, otherwise the system generates alarm A12.6 (spindle tool release cannot perform spindle positioning); When the tool magazine function is effective, the tool magazine is not at the origin position of the spindle cannot be positioned, otherwise the system generates alarm A12.7 (the tool magazine is not at the origin position and the spindle positioning cannot be performed);

  Manual control of spindle orientation conditions: the system is not in operation, spindle tool clamping state;

  Cancel the spindle positioning operation, perform M18 or perform tapping M29 or perform spindle forward and reverse M3/M4, directional cancellation;

  Manually cancel the orientation operation, when the orientation is not completed, press the [Reset button] or emergency stop alarm or press [Spindle Directional Key] or press [Spindle Forward/Reverse Key] to cancel the positioning when the system is not in operation; When the positioning has been completed, you can only press the [spindle directional key] or [spindle forward/reverse key] to cancel the positioning in the emergency stop alarm or when the system is not running, and press the [reset key] to cancel the positioning.

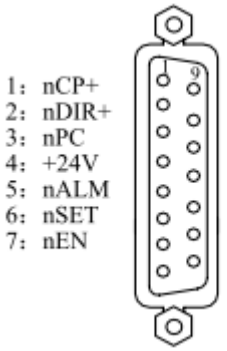
  Note: When the spindle brake output is output, you need to wait for the spindle brake output to be completed before exporting the position mode.

Note: When the bus system is driven by an M3 bus spindle, refer to Appendix XVIII for the orientation function

Note: Please refer to Appendix 18 for the directional function of the feed servo

2.6 Connection of the drive unit

2.6.1 Driver interface definition



Signal	Explain
nCP+、nCP-	Command pulse signal
nDIR+、nDIR-	Command direction signal
nPC	Zero signal
nALM	Drive alarm signal
nEN	Axis enable signal
nSET	Pulse stop signal

CN11, CN12, CN13**, CN14 (15-pin hole) interfaces**

Note: nALM axis alarm signal is valid with 0V; nPC zero signal is valid with 24V.

2.6.2 Axis control enable

Relevant parameters:

Status parameters:

01 3.5==0 Each axis is enabled by the G39 signal (0:No 1:Yes).

0	1	3

****	****	BIT5	BIT4	****	****	****	****

BIT5 : Each axis is enabled via G39 signal (0:no 1:yes).

Note 1: When it is 0, the axis control enable is always issued;

Note 2: When it is 1, the axis control can be triggered by the G39 signal and controlled by PLC;

PLC parameters: (valid when 014.5=1).

K025.0==0 Emergency Stop (0:Yes 1:No) turns off X-axis enable

K025.1==0 Emergency Stop (0:Yes 1:No) turns off Z-axis enable

K025.2==0 Emergency Stop (0:Yes 1:No) turns off Y-axis enable

K025.3==0 Emergency Stop (0:Yes 1:No) turns off 4-axis enable

K025.4==0 Emergency Stop (0:Yes 1:No) turns off 5-axis enable

K	2	5

BIT7	BIT6	BIT5	BIT4	BIT3	BIT2	BIT1	BIT0

BIT0 : Emergency Stop (0:Yes 1:No) Off X-axis enable

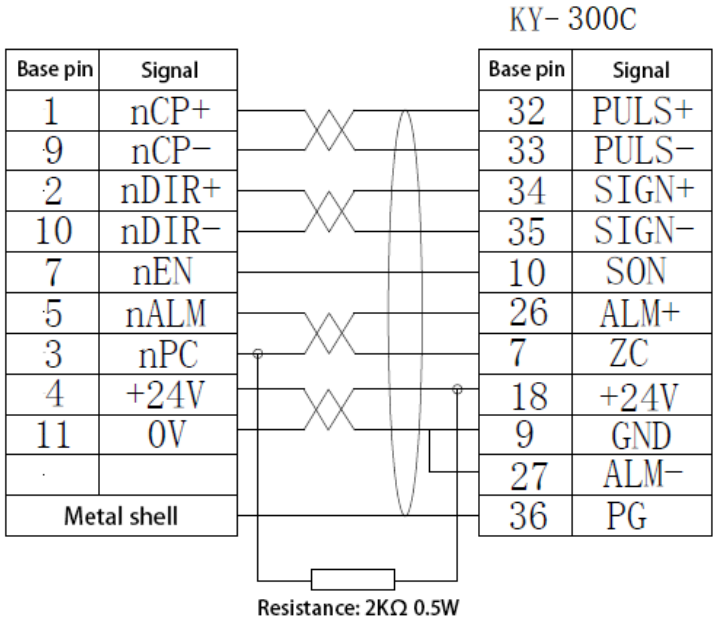
BIT1 : Emergency Stop (0:Yes 1:No) turns off Z-axis enable

BIT2 : Emergency Stop (0:Yes 1:No) turns off Y-axis enable

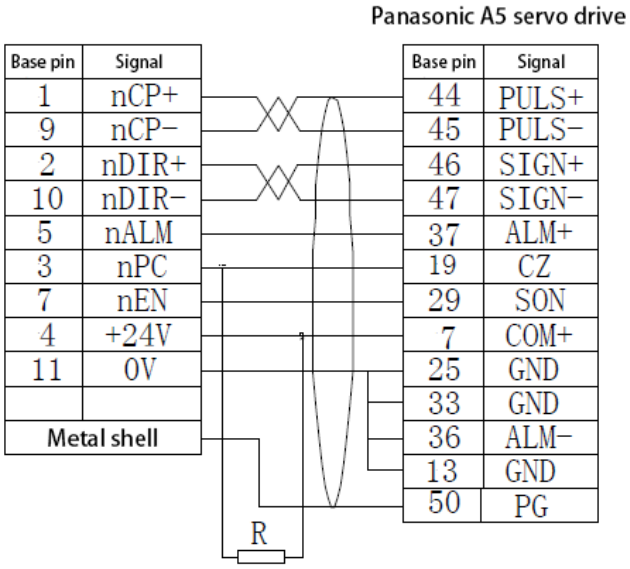
BIT3 : Emergency Stop (0:Yes 1:No) Off 4-axis enable

BIT4 : Emergency Stop (0:Yes 1:No) Off 5-axis enable

2.6.3 Connection diagram of the system to the KY-30 0C drive



2.6.4 Connection diagram of the system and Panasonic A5 driver:



Note: Since the return reference point signal PC is active at high level and the Panasonic Z-phase output signal CZ is low level, a resistor should be connected in parallel (resistance is 2.2KΩ 0.5W).

Note: Resistor R is 2K/0.5W;

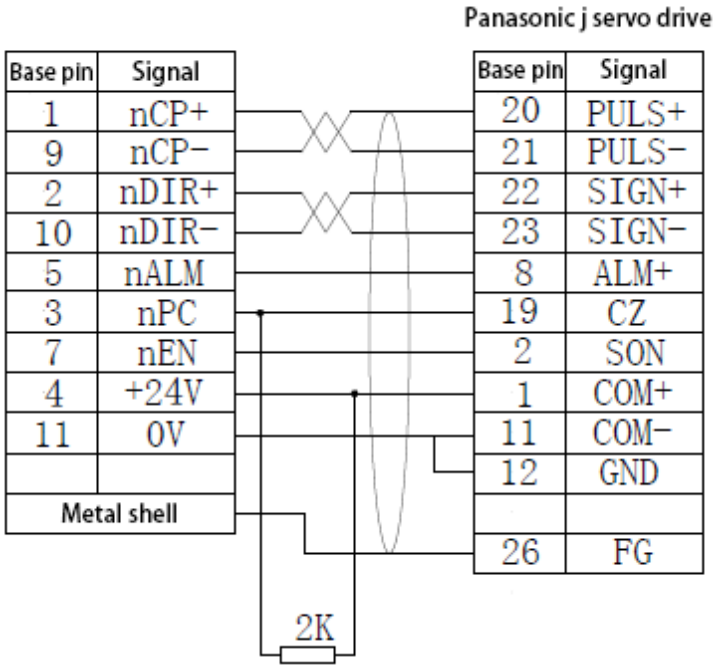
Drive parameters:

Pr0.00 Direction of rotation

PR0.01 control mode setting (0: position mode).

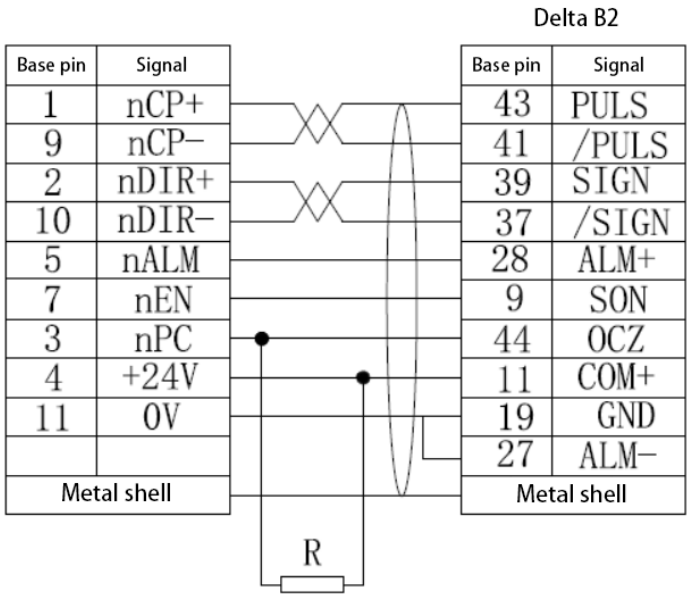
Pr0.05=1 command pulse input to select 0--1, select optocoupler or long-line drive input; (long drive for high speed, optocoupler for low speed) 44/45, 46/47 with long drive; 3/4,5/6 with low speed;
PR0.06 command pulse rotation direction setting;
PR0.07 Command pulse input mode setting 0--3,3: pulse + direction;
Pr0.08 Number of command pulses per 1 revolution;
PR0.09 command pulse molecule;
Pr0.10 command pulse denominator;
Pr1.00 position loop gain;
Pr1.01 velocity loop gain;
Pr1.02 velocity ring integral time constant;

2.6.5 Connection diagram of the system and Panasonic J-series



Note: Resistor R is 2K/0.5W;

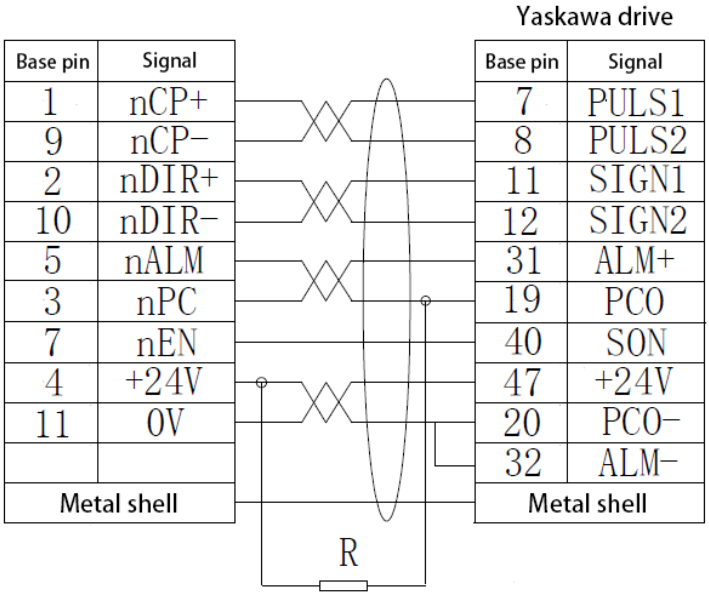
2.6.6 Connection diagram between the system and Delta B2:



Note: Since Delta driver's one-turn signal output is low and the system receives input signal is high, a 2.2KΩ 0.5W resistance is required to parallel the drive CZ output.

Note: Resistor R is 2K/0.5W;

2.6.7 Connection diagram between the system and Yaskawa 7 series:



Note: Resistor R is 2K/0.5W;

Drive parameters:

PN000 = N0010 Position control mode (1 bit is the control mode).

PN100 velocity loop gain

PN101 velocity ring integral constant

PN102 position loop gain

PN200 = N0000 Position Mode Pulse Receive Mode (Pulse + Direction Positive Logic).

= n0005 Position mode pulse receive mode (pulse + direction negative logic [direction negation]).

Pn20E = electronic gear ratio molecule

PN210 = Electronic gear ratio denominator

Motor encoder resolution: determined according to motor model; 7 series motor 2 to the 24th power = 16777072

Chapter III: Description of Common Parameters

3.1 Programming operation parameters

Status parameter

0	0	1	****	****	****	BIT4	BIT3	BIT2	****	****

BIT3 : (0: single step 1: handwheel) mode

BIT4 : Spindle speed (0: analog voltage control 1: switching control).

0	0	3	****	****	****	BIT4	****	BIT2	****	****

BIT2 : Manual removal of mechanical coordinates (0:none 1:Yes) is valid

0	0	4	****	****	****	BIT4	****	****	****	BIT0

BIT6 : Minimum moving unit 0: 0.001,0.0001 1: 0.0001,0.00001 (mm/deg,inc)

0	0	5	****	BIT6	BIT5	BIT4	BIT3	****	****	BIT0

BIT3 : M02 begins with the cursor (0: does not return 1: return).

BIT4 : M30 cursor (0:No 1:Yes) returns to the beginning

BIT6 : Program editing (0:Yes 1:No) supports block deletion

0	1	0	****	****	BIT5	BIT4	BIT3	BIT2	BIT1	BIT0

BIT4 : Arc programming maximum speed limit (0: invalid 1: valid).

0	1	1	****	BIT6	****	****	****	****	BIT1	BIT0

BIT1 : Program setting whether the switch is on when booting (0:Yes 1:No).

BIT6 : System power on (0:Yes 1:No) to set the spindle speed

0	1	2	****	BIT6	BIT5	BIT4	BIT3	BIT2	BIT1	BIT0

BIT0 : Fast movement before zero (0:neither1:Yes) works

BIT1 : Read-ahead cutting feed acceleration and deceleration smoothing (0: invalid 1: valid).

BIT2 : High-speed machining (0:no 1:Yes) on

BIT4 : Acceleration and deceleration during JOG operation (0: linear 1: stepped).

BIT5 : Read-ahead mode with step speed smoothing (0: invalid 1: valid).

0	1	3	****	BIT6	BIT5	BIT4	BIT3	BIT2	BIT1	BIT0

BIT0 : The coordinates increase when the handwheel (0:inverse 1:clockwise) is selected

BIT1 : Number of M99 pieces performed (0:No 1 :Yes) plus 1

BIT2 : Press the reset key to reset the total number of pieces completed (0: not cleared 1: zero).

BIT3 : Read file format type (0: generic 1: carving).

BIT5 : Axis enable signal (0:no 1:Yes) is controlled by PLC

BIT6 : Whether to use the electronic handwheel test cut function (0:no 1:Yes).

BIT7 : Whether the rotation displacement of the handwheel disc is fully run (0:no 1:Yes)

0	1	4	****	BIT6	BIT5	BIT4	BIT3	BIT2	BIT1	BIT0

BIT1 : Fast feed, when the fast feed multiplier is Fo (0: no stop 1: stop).

BIT3 : Back-off function after rigid tapping stops (0: valid 1: invalid).

BIT4 : G73-G89 commands manual intervention to move other axes after interpolation is complete (0:Yes 1:No).

BIT6 : Whether the maximum control speed of the cutting feed is independent for each axis (0:no 1:yes)

BIT7 : Cutting feed rate (0: read program F value 1: internally given).

0	1	5	****	****	****	****	****	BIT2	BIT1	BIT0

BIT0 : Mechanical zero point (0: no memory 1: memory).

BIT1 : G28 command when reference point is not established (0: use stop 1: alarm).

BIT2 : Run the G28 external command without returning zero (0: no alarm 1: alarm).

BIT3 : The reference point is manually returned to the reference point after the memory is established as (0:fast 1:manual) speed

BIT6 : Perform G28 (0:No1:Yes) Use the stop every time

BIT7 : Manual zeroing (0:Yes 1:No) selects multiple axes simultaneously

0	1	8	****	****	****	****	BIT3	****	BIT1	BIT0

BIT0 : Macro program public variable #100~#199, after reset (0: not cleared 1: cleared).

BIT1 : Macro program local variable #1~#50, after reset (0: not cleared 1: cleared).

BIT3 : Whether the single part machining time cutting is automatically cleared to zero (0:no 1:Yes)

BIT4 : Machining time whether the cutting power off is cleared (0:no 1:Yes)

0	1	9	BIT7	BIT6	BIT5	BIT4	BIT3	BIT2	BIT1	BIT0

BIT2 : Radius compensation interference check (0:no 1:Yes)

BIT3 : Starting and retracting forms in tool radius compensation (0:A type, 1:B type).

BIT4: G28, G30 command to move to the middle point, (0: do not cancel 1: cancel) radius compensation

BIT5 : G28, G30 move to the midpoint, cancel knife time (0: standard action 1: vertical).

BIT6 : Tool radius compensation amount (0: radius value 1: diameter value) setting

BIT7 : Radius compensation, whether the corner arc function is valid (0:no 1:Yes)

0	2	3	****	****	****	BIT4	BIT3	BIT2	BIT1	BIT0

BIT1 : Tool setter function (0: valid 1: invalid).

BIT2 : Tool setting instrument function (0:1 knife 1:Multiple knives).

0	2	5	****	****	****	BIT4	BIT3	BIT2	BIT1	BIT0

BIT0 : Location & monitoring display (0: relative coordinates 1: remaining movement).

BIT1 : Press the [Edit] key to switch to the program interface (0:No 1:Yes).

BIT2 : Whether to switch to the alarm interface when an alarm occurs (0:no 1:Yes).

BIT6 : Press the reset button to return to the beginning of the program in (0:Edit 1:Other) mode

0	2	9	****	****	BIT5	BIT4	BIT3	BIT2	BIT1	BIT0

BIT4 : Whether to prohibit program editing with program numbers 8000~8999 (0:No 1:Yes)

BIT5 : Whether to prohibit program editing with program number 9000~9999 (0:No 1:Yes)

0	3	1	****	****	BIT5	BIT4	BIT3	BIT2	BIT1	BIT0

BIT0 : Rigid tapping pitch F setting (0:No 1:Yes)

0	4	2	BIT7	****	BIT5	BIT4	BIT3	BIT2	BIT1	BIT0

BIT0 : Whether to use constant surface cutting speed control (0:No 1:Yes)

BIT3 : G68 coordinate rotation rotation angle (0: absolute command 1: G90/G91 command).

BIT7 : The Z axis moves in a (0:G00 1:G01) manner during program restart

0	4	3	BIT7	****	BIT5	BIT4	BIT3	BIT2	BIT1	BIT0

BIT0 : Whether to use the zoom function (0:No 1:Yes).

BIT2 : Whether the X-axis scaling is valid (0:No 1:Yes).

BIT3 : Whether Y-axis scaling is valid (0:no 1:Yes).

BIT4 : Whether Z-axis scaling is valid (0:no 1:Yes).

BIT7 : Specify the zoom ratio of each axis (0: P command for each axis 1: IJK command for each axis).

0	4	3	BIT7	****	BIT5	BIT4	BIT3	BIT2	BIT1	BIT0

BIT3: G76, G87 displacement (0:Q command 1:I,J,K command)

BIT4 : Set the G76, G87 retract direction (0:positive 1:negative).

BIT5 : Set the G76, G87 retract axis (0:X axis 1:Y axis).

BIT7 : Deep hole drilling (G73, G83), when no cut amount is commanded (0: no alarm 1: alarm).

0	4	3	BIT7	****	BIT5	BIT4	BIT3	BIT2	BIT1	BIT0

BIT0 : When the power is turned on or the state is cleared (0:G00 mode 1:G01 mode).

BIT1 : Whether the plane selection is G17 when power on or clear state (0:No 1:Yes).

BIT2 : Whether the plane selection is G18 when power is on or clear state (0:No 1:Yes).

BIT3 : Whether the plane selection is G19 when power on or clear state (0:No 1:Yes).

BIT4 : Set when power is turned on or cleared (0:G90 mode, 1:G91 mode).

BIT5 : Set when power is turned on or when state is cleared (0:G12,1:G13).

0	4	3	BIT7	****	BIT5	BIT4	BIT3	BIT2	BIT1	BIT0

BIT0 : Do not allow the input of tool wear bias via MDI (0:No 1:Yes)

BIT1 : Whether to disable input of tool geometry offset via MDI (0:No 1:Yes).

BIT2 : Whether to disable input of macro variables via MDI (0:No 1:Yes).

BIT3 : Whether to prohibit the input of workpiece origin bias via MDI (0:No 1:Yes).

0	5	0	****	****	****	****	BIT3	BIT2	****	BIT0

BIT0 : Whether to automatically insert sequential numbers (0:no 1:Yes).

BIT1 : Start behavior when the program status interface executes (0: first line 1: line where the cursor is located).

BIT2 : MDI interface after executing the program (0:no 1:Yes) deletes the program

BIT3 : Press the reset button (0:No 1:Yes) to delete the program under the MDI interface

0	5	3	BIT7	BIT6	****	****	BIT3	BIT2	****	BIT0

BIT6 : Relative coordinate system after reset (0: do not cancel 1: cancel).

BIT7 : Zero relative coordinates (0:no 1:Yes) canceled

Data parameters

0	8	3	Cutting feed rate when power is turned on
0	8	4	The cutting feed is internally given speed
1	1	9	Arc maximum programmed speed limit (default: 5000)
3	1	6	The increment value of the number of sequential numbers is automatically inserted
3	3	6	Minimum pause time at the bottom of the hole
3	3	7	Maximum pause time at the bottom of the hole
3	7	4	The spindle speed value is set at the system power-up

3.2 Feed shaft parameters

Status parameter

0	0	8	****	****	****	BIT4	BIT3	BIT2	BIT1	BIT0

BIT0 : The direction signal is high when the X axis (0:negative 1:positive) is moving

BIT1 : The direction signal is high when the Y-axis (0:negative 1:positive) is moving

BIT2 : Z axis (0:minus 1:positive) is high when the direction signal is moving

BIT3 : 4th axis (0:negative 1:positive) direction signal is high when moving

BIT4 : 5th axis (0:minus 1:positive) direction signal is high when moving

0	0	9	BIT7	****	****	BIT4	BIT3	BIT2	BIT1	BIT0

BIT0 : X-axis driver alarm signal is (0:high 1:low) level alarm

BIT1 : Y-axis driver alarm signal is (0:high 1:low) level alarm

BIT2 : Z-axis driver alarm signal is (0:high 1:low) level alarm

BIT3 : 4th axis driver alarm signal is (0:high 1:low) level alarm

BIT7 : The spindle alarm signal is (0:high 1:low) level alarm

0	2	0	****	****	****	BIT4	BIT3	BIT2	BIT1	BIT0

BIT0 : X axis movement direction button (0: negation 1: no negation).

BIT1 : Y axis movement direction button (0: negate 1: do not negate).

BIT2 : Z axis movement direction button (0: negation 1: no negation).

BIT3 : 4th axis movement direction button (0: negate 1: do not negate).

BIT4 : 5th axis movement direction button (0: negation 1: no negation).

0	3	3	****	****	BIT5	****	BIT3	BIT2	BIT1	BIT0

BIT0 : X-axis is set to (0: linear axis 1: rotation axis).

BIT1 : Y axis is set to (0: linear axis 1: axis of rotation).

BIT2 : Z axis is set to (0: linear axis 1: axis of rotation).

BIT3 : Set the 4th axis to (0: linear axis 1: rotation axis).

BIT4 : Set the 5th axis to (0: linear axis 1: rotation axis).

0	3	5	****	****	BIT5	BIT4	BIT3	BIT2	BIT1	BIT0

When BIT0 : 4th axis is the axis of rotation, absolute coordinate loop function (0: invalid 1: valid).

BIT1 : When the 4th axis is the axis of rotation, (0: rotate in the direction of the symbol 1: rotate nearby).

When BIT2 : 4th axis is the axis of rotation, relative coordinate loop function (0: invalid 1: valid).

0	3	7	****	****	BIT5	****	BIT3	BIT2	BIR1	BIT0

When the BIT0 : 5th axis is the axis of rotation, the absolute coordinate loop function (0: invalid 1: valid).

BIT1 : When the 5th axis is the axis of rotation, (0: rotate in the direction of the symbol 1: rotate nearby).

BIT2 : 5th axis is the axis of rotation, relative coordinate loop function (0: invalid 1: valid).

Data parameters

0	9	0	X-axis G0 fast positioning speed (default: 5000).
0	9	1	Y-axis G0 fast positioning speed (default: 5000).

0	9	2	Z-axis G0 fast positioning speed (default: 5000).
0	9	3	4TH AXIS G0 FAST POSITIONING SPEED (DEFAULT: 5000).
0	9	5	5TH AXIS G0 FAST POSITIONING SPEED (DEFAULT: 5000).

0	8	8	The highest control speed in read-ahead mode (universal to all axes) (default: 8000).

0	8	9	The minimum control speed in read-ahead mode (common to all axes) (default: 0).
0	8	4	The cutting feed is internally given speed

0	8	5	Fo speed for fast running magnification of each axis (common to all axes) (default: 30).

1	1	3	X-axis manual fast positioning speed (default: 5000).
1	1	4	Y-axis manual fast positioning speed (default: 5000).
1	1	5	Z-axis manual fast positioning speed (default: 5000).

1	1	6	4TH AXIS MANUAL FAST POSITIONING SPEED (DEFAULT: 5000).
1	1	7	5TH AXIS MANUAL FAST POSITIONING SPEED (DEFAULT: 5000).

1	1	0	Manual (JOG) continuous feed speed for each axis (default: 2000).

0	7	0	X-axis zeroing high-speed speed (default: 4000).
0	7	1	Y-axis zeroing high-speed speed (default: 4000).

0	7	2	Z-axis zeroing high-speed speed (default: 4000).
0	7	3	4TH AXIS RETURN TO ZERO HIGH SPEED SPEED (DEFAULT: 4000).
0	7	4	5TH AXIS RETURN TO ZERO HIGH SPEED SPEED (DEFAULT: 4000).

0	8	0	Full shaft return to zero low speed speed (default: 80).

0	3	1	The amount of clearance compensation in the X-axis direction (default value: 0).
0	3	2	The amount of clearance compensation in the Y-axis direction (default value: 0).

0	3	3	The amount of clearance compensation in the Z direction (default value: 0).
0	3	4	4TH AXIS DIRECTION CLEARANCE COMPENSATION AMOUNT (DEFAULT: 0).
0	3	5	5TH AXIS DIRECTION CLEARANCE COMPENSATION AMOUNT (DEFAULT: 0).

3.3 Acceleration and deceleration time parameters

Data parameters

1	2	5	S-type time constant for fast X-axis front acceleration and deceleration (default: 100).
1	2	6	S-type time constant for fast Y-axis front acceleration and deceleration (default: 100).
1	2	7	S-type time constant for rapid Z-axis forward acceleration and deceleration (default: 100).

1	2	8	Fast 4-axis front acceleration and deceleration S-type time constant (default: 100).
1	2	9	Fast 5-axis front acceleration and deceleration S-type time constant (default: 100).

2	4	3	High-speed machining corner speed (default: 300)
2	4	4	High-speed machining acceleration (default: 500)
2	4	5	High-speed machining efficiency level (default: 0)

2	4	6	Number of high-speed machining optimizations (default: 15)

2	1	5	Stepped acceleration and deceleration time constant for JOG feed per axis (decreases when the motor vibrates)
1	0	2	Maximum clamping speed for incomplete handwheel operation mode (default: 20 00).
1	0	3	Handwheel incomplete running mode acceleration clamping constant (default: 50).
1	0	4	Handwheel linear acceleration and deceleration time constant (default: 120).
1	0	5	Handwheel exponential acceleration and deceleration time constant (default: 80).

3.4 Gear ratio parameters and calculation

Data parameters

0	0	0	X-axis pulse output multiplication system [numerator] (default: 1).
0	0	5	X-axis pulse output frequency division system [denominator] (default: 1).

0	0	1	Y-axis pulse output multiplication system [numerator] (default: 1).
0	0	6	Y-axis pulse output frequency division system [denominator] (default: 1).

0	0	2	Z-axis pulse output multiplication system [molecule] (default: 1).
0	0	7	Z-axis pulse output divider system [denominator] (default: 1).

0	0	3	4-axis pulse output multiplication system [molecule] (default: 1).
0	0	8	4-axis pulse output frequency division system [denominator] (default: 1).

0	0	4	5-axis pulse output multiplication system [molecule] (default: 1).
0	0	9	5-axis pulse output frequency division system [denominator] (default: 1).

Gear ratio calculation for 2500 wire speed motor direct screw:

$$\frac{\text{Numerator}}{\text{Denominator}} = \frac{10}{\text{Screw pitch}}$$

2500 wire speed motor connection lead screw with gear ratio calculation:

$$\frac{\text{Numerator}}{\text{Denominator}} = \frac{10}{\text{Screw pitch}} \times \frac{Z_M}{Z_D}$$

The 2500 line speed motor calculates the gear ratio of the rotating (indexing) shaft:

$$\frac{\text{Numerator}}{\text{Denominator}} = \frac{1}{36} \times \frac{Z_M}{Z_D}$$

Note: ZM: The number of teeth of the lead screw end gear

ZD: The number of teeth of the motor end gear

Chapter IV Description of other functions debugging

4.1 Spindle orientation function/tapping function debugging

Diagnostic signals (defined by standard PLC programs).

Signal	The spindle is oriented into place	Spindle directional output	Spindle position mode
Diagnostic address	X4.2	Y4.2	Y4.0
Interface pins	CN15.08	CN15.22	CN15.20
remark		M19/M18	M29/M28

Note: The X4.2 signal is valid for +24V on or off

Control parameters

PLC parameters

The K021.1 spindle positioning completion signal is (0:NO 1:NC) active

Note: With 6-axis hardware interface or bus version system new 6-axis tapping option

Tap to select the axis

Parameter 5.0 [tapping selection (0: 5th axis 1: 4th axis)].

Parameter 5.2 [tapping selection 6th axis (0: invalid 1: valid)].

Parameter 5.7 [tapping selection 7th axis (0: invalid 1: valid)].

When selecting the 6th axis is valid, selecting the 4/5 axis is invalid;

When selecting the 7th axis is valid, selecting the 4/5/6 axis is invalid;

Status parameter

001.4==0 Spindle speed (0: analog voltage control 1: switching control).

005.0==0 Rigid tapping selection 4th axis (0:no 1:Yes) [When 0 is selected, pulse tapping is sent with the pulse hole of the 5th axis; When 1 is selected, pulse is sent out at the 4th axis pulse port].

031.0==1 Pitch F setting for rigid tapping (0:No 1:Yes) [When 0 is selected, the tapping section is programmed for speed F.] = spindle speed * tap pitch; When 1 is selected, the tapping section is programmed with speed F value = tap pitch].

058.0==1 The spindle control mode is (0:Follow 1:Servo) [follow is flexible tapping, servo is rigid tapping].

058. 1==0 Whether the tapping becomes a high-speed deep hole tapping cycle (0:No 1:Yes) [When the high-speed deep-hole tapping cycle is not selected, when editing G84Q_, the Z-axis retract retreats to point R.] , and then tap back and forth; When the high-speed tapping cycle is selected, when editing the G84Q_, the Z-axis retract distance is the parameter set value (number parameter 339 [The amount of backoff during the deep hole tapping cycle (default: 2)]) is then reciprocated and tapped.

058.6==0 Whether the magnification is valid when rigid tapping and retracting (0:no 1:Yes) [When the retract magnification is invalid, the tapping and retracting speeds are the same; When the retract ratio is selected, the tapping and retracting speed is subject to the number parameter 3 38 (the magnification value when rigid tapping and retracting (retract speed = F* value of the approach speed)), and the number parameter 338=1, The retract speed is the same as the approach speed, and when the number parameter 338=1.2, the retract speed is 1.2 times the approach speed].

0	0	1	****	****	****	BIT4	****	****	****	****

BIT4 : Spindle speed (0: analog voltage control 1: switching control).

0	0	5	BIT7	****	****	****	BIT3	BIT2	****	BIT0

BIT0 : Rigid tapping selects 4th axis (0:No 1:Yes)

0	3	1	BIT7	****	****	****	BIT3	BIT2	****	BIT0

BIT0 : Pitch F setting for rigid tapping (0:No 1:Yes)

0	5	8	****	****	****	****	****	****	****	BIT0

BIT0 : The spindle control mode is (0:follow 1:servo) [follow for flexible tapping, servo for rigid tapping].

Data parameters

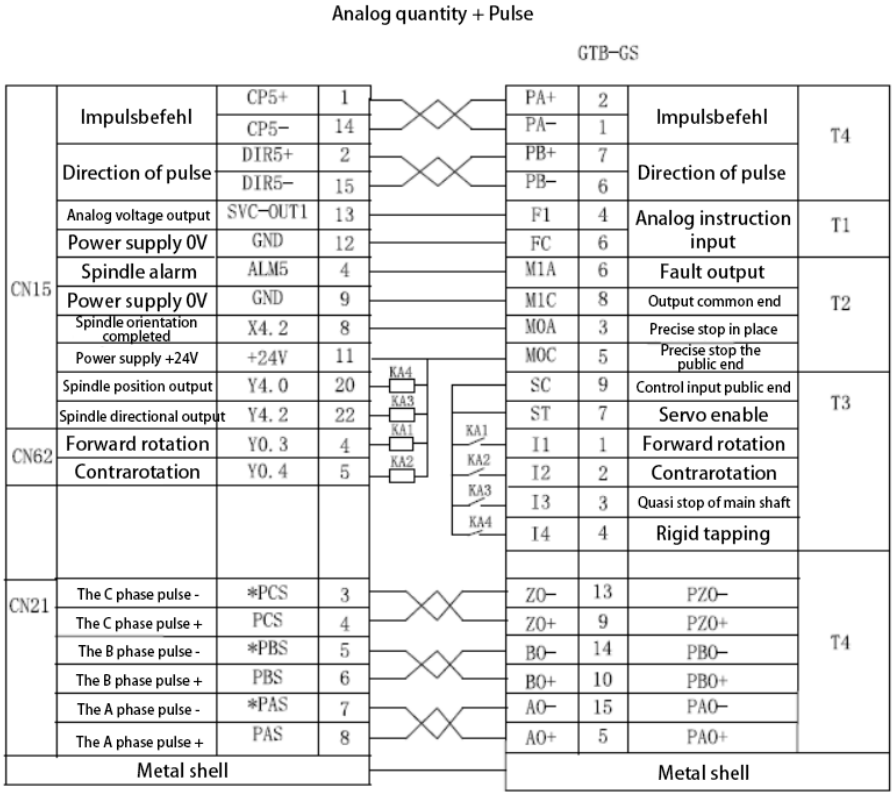
2	3	0	Tapping spindle command multiplication coefficient CMR (tapping pulse molecule) (1st gear gear).
2	3	1	Tapping spindle command multiplication factor CMR (tapping pulse molecule) (2nd gear gear).
2	3	2	The tapping spindle commands the multiplication factor CMR (tapping pulse molecule) (3rd gear gear).
2	3	3	The tapping spindle commands the frequency division factor CMD (tapping

			pulse denominator) (1st gear gear).
2	3	4	The tapping spindle commands the frequency division factor CMD (tapping pulse denominator) (2nd gear gear).
2	3	5	The tapping spindle commands the frequency division factor CMD (tapping pulse denominator) (3rd gear).

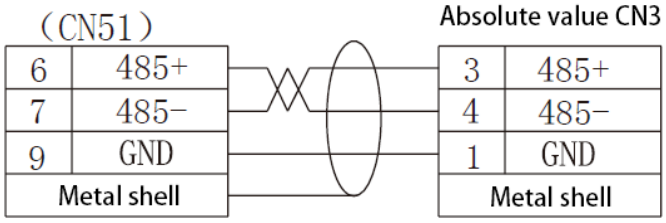
Note: 2500 line number motor 230=10;233=1;1024 line number motor 230=512;233=125;

Note: 230=10; 233=1 when the system defaults to spindle rotation of 10,000 pulses, other gear ratios can be set in the drive;

Wiring reference diagram



4.2 With K300C absolute value driven debugging



The milling machine series is connected to the KY300C absolute actuation communication

Debugging steps:

1、 Check the communication cable connection first.

2、 Drive and motor adjustment:

Motor zeroing steps:

1） Modify the drive parameter PA-0 to 620, change PA-1 to (motor signal corresponds to motor code), return Wr (looks like ur), press 【Enter】3 seconds, "Finish" is displayed. Restart the drive to see if the PA-67 is close to the motor rated current multiplied by 10.

3、 Drive parameter adjustment:

Drive parameter adjustment steps:

1） Eliminate drive alarm: If the drive reports alarm No. 42, you need to change PA-0 to 620, PA-1 to (corresponding to the motor code), return to WR mode and press [Enter] for more than 3 seconds until the "Finish" operation is successful, and the power failure takes effect; "Error" is a failure or incorrect password) [WR mode looks like ur].

If the power on drive alarm number 40 change PA-99 to 1.

If the driver calls police number 44, change PA-0 to 620; PA-4 changed to 4; PA-53 changes to 0001, returns to co--mode, and presses [Enter] until A.2000 is displayed; Then return to PA-0 and change to 510.

2) Set the drive parameters: first set **the** motor code, change PA-0 to 620, PA-1 to (corresponding to the motor code).

Modify PA-0==510 again, and change PA-80 to (?). (The X axis corresponds to 1, the Z axis corresponds to 2, the Y axis corresponds to 3, and the 4th axis corresponds to the corresponding axis.) 4. The 5th axis corresponds to 5);

(PA81==2, PA-82==0, PA-84==0 driver defaults do not need to be changed).

3) Drive parameter saving: return to EE - mode, press [Enter] to enter EE-SET mode, and hold [Enter] until "appears" Finish" is complete.

4、 System parameter modification:

To modify the status parameter:

003.2==1 Manual clearing of mechanical coordinates is valid
004.0==0 Axis 1 (0:Yes 1:No) reads the bus (X-axis).
004.1==0 Axis 2 (0:Yes 1:No) reads the bus (Y-axis).
004.2==0 3rd axis (0:Yes 1:No) reads the bus (Z-axis).
004.3==0 4th axis (0:Yes 1:No) reads the bus (4th axis).
004.4==0 5th axis (0:Yes 1:No) reads the bus (5th axis).

015.0==1 Mechanical zero point memory;
006.5==1 Communication mode bus
022.1==1 The soft limit before returning to mechanical zero is valid
022.2==1 When the overtravel command is issued, alarm is issued after overtravel
022.3==0 The zero return method soft limit is valid

051.0==1 Stroke detection before moving

051.1==0 Reserved

051.2==1 Fast moving soft limit alarm (0: lag 1: accurate).

Note: 004.0~004.4 Change to 0 when using absolute values for each axis

0	0	3	****	****	****	****	****	BIT2	****	****

BIT2 : Manually clear mechanical coordinates (0: invalid 1: valid);

0	0	4	****	****	****	BIT4	BIT3	BIT2	BIT1	BIT0

BIT0 : Axis 1 (0:Yes 1:No) is the read bus (X-axis).

BIT1 : Axis 2 (0:Yes 1:No) is the read bus (Y-axis).

BIT2 : Axis 3 (0:Yes 1:No) is the read bus (Z-axis).

BIT3 : Axis 4 (0:Yes 1:No) is the read bus (4-axis).

BIT4 : Axis 5 (0:Yes 1:No) is the read bus (5-axis).

0	1	5	****	****	****	****	****	****	****	BIT0

BIT0 : Mechanical zero point (0: no memory 1: memory).

0	2	2	****	****	****	****	BIT3	BIT2	BIT1	BIT0

BIT1 : Soft limit before zero return (0: invalid 1: valid).

BIT2 : When the overtravel command is issued, alarm is issued in overtravel (0:front 1:after).

BIT3 : Zero return mode soft limit (0:no 1:Yes) is valid

0	5	1	****	****	****	****	BIT2	****	****	BIT0

BIT0 : Stroke detection before moving (0:no 1:Yes).

BIT1 : Fast Moving Soft Limit Alarm (0: Hysteresis 1: Accurate).

To modify data parameters:

3	0	1	The baud rate of serial communication (default: 19200) **is changed to 19200**
1	5	9	Link mode coordinate alarm range (default value: 0 1) **Change to 0.1**
2	4	7	The absolute value (0:KY1000 1:SANKYO 2:98D) (default: 0) **is 0**
2	4	8	The absolute stop bit (0: odd check 1: no check) (default: 0) **is 0**

5、 Electronic gear ratio setting

System data parameters:

0	0	0	X-axis pulse output multiplication system [numerator] (default: 1).
0	0	5	X-axis pulse output frequency division system [denominator] (default: 1).
0	0	1	Y-axis pulse output multiplication system [numerator] (default: 1).

0	0	6	Y-axis pulse output frequency division system [denominator] (default: 1).

0	0	2	Z-axis pulse output multiplication system [molecule] (default: 1).
0	0	7	Z-axis pulse output divider system [denominator] (default: 1).

0	0	3	4-axis pulse output multiplication system [molecule] (default: 1).
0	0	8	4-axis pulse output frequency division system [denominator] (default: 1).

0	0	4	5-axis pulse output multiplication system [molecule] (default: 1).
0	0	9	5-axis pulse output frequency division system [denominator] (default: 1).

Absolute gear ratio algorithm:

The number of pulses emitted by the encoder in one turn (10000) ÷ (L×1000); L is the screw pitch; When there is a reduction ratio, it is multiplied by the reduction ratio.

6、 Machine zero-clearing:

Press the [Location] button continuously until the [Current Location (Comprehensive Coordinates)] interface is displayed in the upper left corner of the system screen;

Press the [Machine Zero Point] key, and then press the system panel letter [X/Z/Y/A/C] key, X, displayed on the machine coordinates

Z, Y, A, C flash and press the [Cancel] key to clear the machine coordinates.

7、 Test run each axis

Move each axis manually or with a handwheel, shut down and restart.

Observe whether the system interface has alarms (the **drive coordinates of each axis are inconsistent with the system coordinates, press the reset button to update or communication errors**).

If there is no alarm, it means normal;

If there is a **prompt** "communication **error alarm"**, check whether the communication line is connected normally, system parameter status parameter 174.0==1 communication mode bus, data parameters: P159==0.1, P301 == 19200 , whether it is set correctly; Whether the drive parameter PA-80 corresponds to the correct axis.

If there is a prompt **"**The driving coordinates of each axis **are inconsistent with the system** coordinates, press the reset button to update the alarm**"**, then the current position (comprehensive coordinates) record the coordinate values of each axis (machine tool coordinates), press the [Reset] key to update the coordinates, and observe whether the coordinate values of each axis of the current position (comprehensive coordinates) (machine tool coordinates) change by more than 0.1.

If the coordinates of individual axes change by more than 0.1, modify the system parameter status parameter 038.0~038.4 (each axis reads the absolute value coordinates (0:positive 1:negative to)); If it is 0, change to 1; If it is 1, change to 0; Turn off the power and restart, press [Reset] and the new coordinates, and then manually or handwheel to move each axis, shut down and restart, if there is no alarm indicates normal; If there is still an alarm, check whether the electronic gear ratio setting is normal.

Note: After adjusting normally, you need to change the direction of axis movement on the system

status parameter 008.

After the above settings are normal, you can set the zero point of the machine tool again and set the soft limit of the machine tool;

Soft limit settings:

System data parameters

0	1	0	X-axis negative maximum travel (default: -9999.9999).
0	1	1	X-axis forward maximum stroke (default: 9999.9999).
0	1	2	Y-axis negative maximum travel (default: -9999.9999).
0	1	3	Y-axis forward maximum stroke (default: 9999.9999).

0	1	4	Z-axis negative maximum travel (default: -9999.9999).
0	1	5	Z-axis forward maximum travel (default: 9999.9999).
0	1	6	4th axis negative maximum stroke (default: -9999.9999).
0	1	7	4th axis forward maximum travel (default: 9999.9999).
0	1	8	5th axis negative maximum stroke (default: -9999.9999).
0	1	9	5th axis forward maximum travel (default: 9999.9999).

4.4 Single Segment of the Program

A single section of the program is valid in the automatic and input mode;

4.5 Program Skipping

When the program skip signal is valid, the program segment marked with "/" at the beginning of the segment is skipped and not executed. Program hopping is valid in the automatic, typed mode.

4.6 Machine Lock

Machine locks work in any way. When the program is running, the machine lock state cannot be switched.

K2.0 machine lock key (0: valid 1: invalid).

4.7 Secondary Lock

The auxiliary lock is valid in automatic mode and entry mode.

K2.1 secondary lock key (0: valid 1: invalid).

4.8 Test cut of the handwheel

In automatic effect.

When switching or turning off the handwheel test cutting function during program operation, the system can choose whether to pause K16.5 in the program;

Number parameter No. 149 [maximum speed limit for handwheel test cut].

Parameter 13.6 [Handwheel test cut reverse (0:no 1:Yes) valid]; When the handwheel is selected to be effective when the handwheel is shaken counterclockwise, the system runs in reverse when the program runs

in reverse to the beginning of the current segment; When the selection is invalid, only a clockwise shake of the handwheel is effective.

Parameter 1 8.7 Handwheel test cut follows handwheel rotation speed (0: invalid 1: valid) [change base 153].

Number parameter 153 Handwheel test cut follows the base of the change

4.9 Select Stop

In the automatic, entry mode is valid, press the 

key to make the selection stop button indicator light on, it means to enter the selection stop state; At this time, when the program runs to the M01 instruction, it will be "paused". You need to press the 

key again before the program can continue.

K10.0 [Program running status skip key (0: valid 1: invalid)].

4.10 One-touch G54 function

When pressing [one-key G54 key] in handwheel/manual mode, the current machine coordinates are automatically entered into the G54 coordinate system to achieve tool setting, provided that K20.4 == 1 (one-key G54 function [0 : invalid 1: valid])

4.11 Breakpoint Startup Function

When the program stops unexpectedly in the middle of running, you can press [breakpoint start key] in automatic mode, and then manually control the spindle start, cooling and other modal information, press [cycle start button] to start the program, and the system automatically jumps to the previous section of the program section running.

4.12 Tapping Fallback Function

Parameter 14.3 Backoff function after rigid tapping stops (0: valid 1: invalid).

Number parameter 341 Rigid tapping retract speed F (default 100).

When tapping is in progress, when the emergency stop alarm makes the tap stuck in the workpiece, you can press [tapping retract button] in other ways of mechanical zero mode, and the spindle is linked with the Z axis (at the speed of the number parameter 341 on the Z axis) Return the tap to the R point plane.

4.13 M code retrieval program

M9000 ---- M9999 Adjustable programs O9000 ---- O9999 programs;

4.14 Return to the origin of the workpiece

Parameter parameter 1 5.4 Return to the workpiece origin (0:Yes 1 No) requires the machine to return to zero

Function key action, according to the data parameter 2 No. 17 (back to the workpiece origin Z axis height)

to make corresponding changes, when the number parameter 217 is 0, press the back to the working origin key Z axis first return to the mechanical zero point, and then X Y axis back to the zero point of the current coordinate system (absolute coordinates X0, Y0); When the number parameter 217 is not 0 (parameter range 0~9 999), press the back to working origin key Z axis to return to the mechanical zero point first, and then X Y axis to return to the zero point of the current coordinate system X / Y (absolute coordinates X 0, Y0), and then return to the workpiece origin height set on the Z axis of the current coordinate system (absolute coordinates Z = number parameter 217);

Note that when the 4th and 5th axis functions are turned on, the order of returning to the origin of the workpiece is affected by the one-key zeroing order of the parameter parameter parameter 24.5 [machine tool one-key zeroing order (0:Z-XY45 1:Z4-XY5)]];
If the bit parameter 24.5 selects 0 (Z-XY45), the zero return action is consistent with the above, the Z axis first returns to the mechanical zero point, X Y45 returns to the zero point of the current workpiece coordinate system, and finally passes the number parameter 2No. 17 [back to the Z axis height of the workpiece origin]. Determine whether the Z axis returns to the height value of the workpiece origin; When the number parameter 2No. 17 is 0, the Z axis does not return to the origin value of the workpiece and maintains the mechanical zero point position; When the number parameter 2 17 is not 0, the Z axis returns to the workpiece origin height set by the Z axis of the current coordinate system (absolute coordinates Z = number parameter 217).

If the parameter 24.5 selects 1 (Z4-XY5), the 4th axis and the Z axis act at the same time, the 4th and Z axes first return to the mechanical zero point, X Y5 returns to the zero point of the current workpiece coordinate system, and finally passes the number parameter 2No. 17 [back to the Z axis height of the workpiece origin]. Determine whether the Z axis and the 4th axis return to the height value of the workpiece origin; When the number parameter 2No. 17 is 0, the Z axis and the A axis do not return to the origin value of the workpiece, and the mechanical zero point position is maintained; When the number parameter 2 17 is not 0, the Z axis and the 4th axis return to the workpiece origin height set by the Z axis and the 4th axis of the current coordinate system (absolute coordinates Z = number parameter 217).

4.15 One-touch tool setting function

In the coordinate system interface, in which coordinate system the current cursor is in, press the one-key tool function key to enter the current mechanical coordinates into the workpiece coordinate system where the cursor is located.

In the case of manual tool setting, the value set on the base offset can be automatically cleared after the manual tool setting is completed, parameter bit parameter 2 4.2 [clear the base offset when manually realigning the coordinate system (0:no 1:yes)].

4.16 One-key return to the machine zero point function

When the effective number parameter of the one-key zeroing function is selected 2 4.4=1 [machine tool one-key zeroing function (0:invalid 1: valid)], press the zero button of the machine to switch Mechanical zeroing mode, the menu function key switches to [All zero, X-axis zero, Y-axis zero, Z-axis zero, 4-axis zero 5 axis zero], when pressing the corresponding function key, perform the corresponding axis zeroing; When you press [All to zero], the axis zeroing sequence parameter 24.5 [machine tool one-key zeroing sequence (0:Z-XY45 1:Z4-XY5)] can be controlled by parameters This parameter also controls the order back to the workpiece origin;

If the parameter 24.5 selects 0 (Z-XY45), the Z axis returns to the mechanical zero point first, and after completion, it returns to the X Y45 axis mechanical zero point at the same time;

If parameter 24.5 selects 1 (Z 4-XY5), the Z axis and the 4th axis first return to the mechanical zero point at the same time, and after completion, return to the X Y5 axis mechanical zero point at the same time;

4.17 The feed rate is 0% fast rate adjustment

When the feed rate is adjusted to 0%, you can choose whether the fast multiplier is changed to F0;
K017. 1 [Fast multiplier (0:no 1:Yes) switch F0 at 0% feed magnification].

4.18 Manual fast movement control

Manual quick movement control can be selected in two ways, one is to light the fast move button and open the function (jog), and the other is to press and hold the fast move button to open the function (long press); The PLC parameter can be selected, K21.7 [Turn on manual quick movement control (0:jog 1:long press)].

NOTE: SOME BUTTON PADS DO NOT SUPPORT LONG PRESS FUNCTION, SUCH AS 1 000MC, 3000M-V; The specific use is selected according to the actual button pad.

Note: When the tool magazine debugging function is turned on, manual quick movement has no effect.

Data parameter 113 manual X-axis fast positioning speed
Data parameter 114 manual Y-axis fast positioning speed
Digital parameter 115 manual Z-axis fast positioning speed
Number parameter 116 manual fourth axis fast positioning speed
Data parameter 117 manual fifth axis fast positioning speed

4.19 Special instructions for the coordinate system when using the knife

In the coordinate system interface to the tool or offset interface, there are two cases when using letters and numbers for input, 1 Output Z+10 at the coordinate system G54 position is input or determined, and the bit parameter 3.3=0 [coordinate system zero point offset], the Z axis is translated in the positive direction by 10mm at the current coordinate as the origin of the Z axis of the G54 coordinate system, So the absolute coordinate of the current point after the knife is -10; 2. Output Z+10 at the coordinate system G54 as input or OK, bit parameter 3.3=1[Absolute], the Z axis is shifted in the negative direction by 10mm coordinates at the current coordinates as the G54 coordinate system Z The origin of the axis (on the contrary, Z-10, the origin is shifted in the forward direction), so the absolute coordinate of the current point after the knife is 10;

It is simply understood that when the bit parameter is 3.3=0, when the coordinate system is opposite to the knife Z+10, the current absolute coordinate is -10 as opposed to the input +10; When the bit parameter is 3.3=1, the current absolute coordinate is +10 when the coordinate system is Z+10.

bit parameter 3.3 [enter letters + numbers when the coordinate system is aligned with the knife (0: offset zero point 1: absolute coordinates)].

Appendix I. Parameter Search and Modification

To modify the parameter password: **769816 Press the Settings key + Page Up and Down (F1) key to enter the Change Password page**

1. How to find data parameters and status parameters:

Press the PAR key to enter the parameter interface, " F1 corresponding bit parameter, F2 corresponding number parameter" press the corresponding button to enter the corresponding page, you can press the P Q key + parameter number, and then press the IN key to find the parameter;

Or press the ⇧ , ⇩ , ⇦ , ⇨ key to find the parameter;

2. How to find PLC (K, T, C, D) parameters:

Press the PLC key to enter the PLC - interface, then press F3 to enter the PLC (K parameter page), and then press F3 to enter the PLC parameter page), " F1 corresponding K parameter, F2 corresponding T parameter, F3 corresponding D parameter page, F4 corresponding C parameter page";

You can press the P Q key + parameter number, and then press the IN key to find the parameter;

Or press the ⇧ , ⇩ , ⇦ , ⇨ key to find the parameter;

Appendix II: Alarms/Warnings

A001.5 Tool magazine debugging mode function can not start the program, K30.7 turn off debugging mode

Alarm reason: After using the tool magazine function, turn on the tool magazine debugging function, and the system starts to run the program alarm;

Handling: Press [Reset button] to cancel the alarm, check whether the tool magazine is normal, if normal, close the tool magazine debugging function K30.7=0; If it is abnormal, turn off the tool changer debugging function after troubleshooting.

A001.6 External loop opening time is too long (normally open point) K20.0

Alarm reason: the cycle start button conduction time exceeds 4 seconds, the system alarm;

Treatment: Check whether the external cycle start button of the system is connected to the normally open contact, and check whether the external switch is damaged.

A001.7 External suspension opening time is too long (connected to normally closed point) K20.1

Alarm reason: the on-time of the external pause button exceeds 4 seconds, and the system alarms;

Treatment: Check whether the external pause button of the system is connected to the normally closed contact, and check whether the external switch is damaged.

A002.0 The guard door is not closed and is not allowed to run automatically

Alarm reason: in automatic mode, the protective door is not closed, the program is started, and the system generates an alarm;

Handling: Press [Reset] to cancel the alarm, if the protective door has been closed, detect whether the protective door sensor is damaged or whether the level selection is normal.

A002.1 Tool setter over stroke alarm

Alarm reason: When the overtravel signal of the tool setter is selected, the overtravel signal of the tool setter has an input, and the Z axis moves in the negative direction;

Processing: Press [Reset] to cancel the alarm and make the Z axis move in the positive direction.

A002.2 Set the alarm for the completion of the total number of parts machining

Alarm reason: the number of processed pieces reaches the number parameter 361 The total number of processed pieces is set, and the number of processed pieces is set to indicate the effective K16.3=1 alarm;

Treatment: Press [Reset] to cancel the alarm and manually clear the number of processed pieces.

A002.3 The tool must not be loosened when the spindle is rotating

Alarm reason: loosen the knife during spindle rotation, and the system generates an alarm;

Handling: Press [Reset] to cancel the alarm.

A002.4 When the spindle rotates, the tool clamping in place signal is invalid alarm

Alarm reason: no tool clamping signal was detected in the spindle rotation;

Treatment: Press [Reset] to cancel the alarm, check the installation position of the tool clamping signal switch and whether it is damaged.

A002.5 The spindle must not be started when the tool clamping signal is invalid

Alarm reason: start the spindle when the tool clamping signal is not detected;

Treatment: Press [Reset] to cancel the alarm, check the installation position of the tool clamping signal switch and whether it is damaged.

A002.6 The spindle tool is loosened and the spindle must not be started

Alarm reason: the tool is in the loose state to start the spindle;

Handling: Press [Reset] to cancel the alarm, and the spindle can only be started when the tool is clamped.

A004.1 is not currently in the spindle analog voltage control state and cannot perform the spindle jog function

Alarm reason: When the spindle control is switch control, open the spindle jog function, and the system alarms;

Handling: Press [reset] to cancel the alarm, and the spindle jog function cannot be used when the spindle switch is controlled.

A004.2 M03, M04 code specified error

Alarm reason: In the forward or reverse state of the spindle, the alarm is given when the spindle rotates in the opposite direction of rotation;

Handling: Press [Reset] to cancel the alarm, the spindle must be stopped to run in the opposite direction.

A004.3 M63, M64 code specified error

Alarm cause: In the state of forward or reverse rotation of the second spindle, the alarm is given when the spindle rotates in the opposite direction of rotation;

Handling: Press [Reset] to cancel the alarm, the spindle must be stopped to run in the opposite direction.

A004.4 Spindle shift time is too long

Alarm reason: When the spindle automatic gear shift is effective, execute the shift code M41-M44, and the code alarm is not completed within the time set by T045;

Handling: Press [Reset] to cancel the alarm, check whether the shift signal is installed correctly, and whether the switch is damaged.

A004. 7 Z axis is not at the zero point other axes can not command zero operation, K22.0 optional

Alarm reason: When the Z axis needs to be at the zero point position other axes to perform the zero return operation K22.0=1, the other axis is performed to zero;

Handling: Press [Reset] to cancel the alarm, return the Z axis to the zero point first, and then perform other axis operations.

A008.0 Spindle speed position mode conversion abnormal

Alarm reason: The code alarm is not completed within the T024 setting time after executing the M29 code;

Handling: Press [Reset] to cancel the alarm, check whether the setting time of T024/T025 is correct (T025<T024), and check whether the signal setting is correct after the position mode switching is completed (K21.0).

A008.1 Spindle position speed mode conversion abnormal

Alarm reason: After executing the M28 code, the code alarm is not completed within the T028 setting time;

Handling: Press [Reset] to cancel the alarm and check whether the setting time of T027/T028 is correct (T027<T028).

A008. 2 Insufficient air pressure alarm, level selection K20.2 detection delay T02

Alarm reason: Insufficient air pressure alarm signal detected within the T02 setting time;

Processing: detect whether the air pressure detection device is normal, and whether the detection signal level selection K20.2 setting is correct;

After waiting for sufficient air pressure, press [Reset] to cancel the alarm.

A010.1 The spindle speed is too high and the tool cannot be released

Alarm reason: the spindle stop state is issued, but the spindle has not slowed down to zero, and the loose knife alarm is set when the speed is large D data D126;

Processing: Press [Reset] to cancel the alarm and check the D126 setting value (D126=20); Or delay the spindle after stopping for a period of time and then loosen the knife.

A010.4 Spindle tool clamping detection abnormalities

Alarm reason: After the tool is released, the clamping signal is not detected within the T020 set time;

Handling: Press [Reset] to cancel the alarm, check whether the T020 setting time is correct, check the installation position of the switch and whether the switch is damaged.

A010.5 Spindle tool loosening detection abnormality

Alarm reason: After the tool is released, the signal of loosening in place is not detected within the T09 set time;

Handling: Press [Reset] to cancel the alarm, check whether the T09 setting time is correct, check the installation position of the switch and whether the switch is damaged.

A010.7 Spindle loose knife tightening detection abnormality

Alarm reason: Simultaneous sensing of loose knife/clamp knife in place;

Treatment: Press [Reset] to cancel the alarm, check the installation position of the switch and whether the switch is damaged.

A011.0 Indexing table lock is not in place

Alarm reason: When the clamping/releasing signal is output, the loosening/clamping signal is not detected in place within the time of T043/T044.

Treatment: Press [Reset] to cancel the alarm, check the installation position of the switch and whether the switch is damaged.

A011.1 Indexing table lock loosening detection abnormalities

Alarm reason: Loosen in place and clamp in place spot are detected at the same time.

Treatment: Press [Reset] to cancel the alarm, check the installation position of the switch and whether the switch is damaged.

A011.2 Indexing table is not released non-movable 4th axis

Alarm reason: In the case of automatic indexing work effective K1.7=1, the system does not detect the release signal when moving the 4th axis

Handling: Press [Reset] to cancel the alarm and move the 4th axis in the state of release of the indexing table.

A011.3 The indexing table is not locked and cannot move the XYZ axis

Alarm reason: In the case of automatic indexing work effective K1.7=1, K1.4=0, the system does not detect the signal to loosen into place when moving any XYZ axis

Handling: Press [Reset] to cancel the alarm, move the XYZ axis in the clamping state of the indexing table or change K1.4 to 1.

A012.5 Spindle directional timing detection anomalies

Alarm reason: directional alarm when the spindle is oriented into place signal and there is an input;

Handling: Press [Reset] to cancel the alarm and reset the input level of the spindle directional signal in place (K21.1).

A012.6 Spindle positioning cannot be performed when the spindle tool is released

Alarm reason: spindle directional output alarm when the tool is released;

Handling: Press [Reset] to cancel the alarm, and it cannot be oriented when the tool is released.

A012.7 (bucket pipe, disc) tool changer is not at the origin and spindle positioning cannot be performed
(Turret type) The Z-axis is in the tool change interval and cannot perform spindle positioning

Alarm reason: bucket type tool magazine: the tool magazine is not in the back position, and an alarm is generated when the spindle performs positioning;

Disc tool changer: the ATC tool arm is not at the origin position, and an alarm is generated when the spindle performs positioning;

Turret tool magazine: when the Z axis is in the tool change interval, an alarm is generated when the spindle performs positioning;

Handling: Press [Reset] to cancel the alarm, make sure that the tool magazine returns to its original position, and then perform the spindle positioning operation.

A013.0 command knife number T is 0

Alarm reason: turret tool magazine: Executing the T0 code when changing tools generates an alarm

Handling: Press [Reset] to cancel the alarm, the turret tool magazine cannot execute the T0 code

A013.1 Spindle orientation in place to detect anomalies

Alarm reason: After the system outputs the spindle orientation, the spindle orientation signal is not detected after the T023 and T018 setting time;

Handling: Press [Reset] to cancel the alarm, the spindle orientation signal X4.2 and +24V on/off are effective, check whether the wiring is normal; Check whether the T023/T018 time setting is normal.

A013.2 The spindle tool number is inconsistent with the tool magazine and the tool magazine cannot be advanced

Alarm reason: After turning on the tool magazine function, when the current tool number of the spindle is inconsistent with the tool magazine cutterhead tool number, manually control the tool magazine forward alarm in the manual/handwheel mode;

Handling: Press [Reset] to cancel the alarm, the tool magazine forward condition is that the current spindle tool number is equal to the tool magazine tool number, and the manual rotation tool magazine transposes the required tool number position.

The A013.3 Z-axis is not at the **2/3rd reference point and the tool changer cannot be advanced**

Alarm reason: After turning on the tool magazine function, when the Z axis is not at the 2nd or 3rd reference point, manually control the tool magazine forward alarm in the manual/handwheel mode;

Handling: Press [Reset] to cancel the alarm, the tool magazine forward condition is the Z axis and then the 2nd or 3rd reference point position, and the G91G 30 Z0 can be returned to the number in MDI mode 2 reference points.

The tool changer is not rotatable when the A013.4 (Bucket Pipe) tool changer is not in the set position and the Z-axis is not at the 3rd reference point

The (disc type) tool magazine sleeve is not in place for the tool return, and the tool magazine cannot be rotated

(Turret type) The Z-axis is not in the allowable rotation range of the tool magazine, and the tool magazine cannot be rotated

Alarm reason: control the tool magazine rotation alarm when the Z axis is not at the 3rd reference point or the tool magazine is not in the receding position;

Disc type tool magazine When the tool sleeve is not in place for the tool return, control the tool magazine rotation alarm;

Turret type tool magazine when the Z axis is not in the allowable rotation range of the tool magazine, control the tool magazine rotation alarm;

Handling: Press [Reset] to cancel the alarm.

A013.5 Non-manual/handwheel mode does not allow manual control of the tool changer

Alarm reason: After turning on the tool magazine function, the tool magazine alarm is not manually controlled in the manual/handwheel mode;

Handling: Press [Reset] to cancel the alarm, manual control tool magazine conditions must be controlled in manual/handwheel mode.

A013.6 (bucket type, disc type) tool changer is not at the origin position and cannot return to zero

The (turret type) Z-axis is not in the tool magazine and the rotation interval is not zero

Alarm reason: The bucket type tool magazine is not in the back position, and the control tool magazine returns to zero alarm;

When the disc type tool magazine sleeve is not in the return position, control the tool magazine to zero alarm;

Turret tool magazine: when the Z axis is not in the allowable rotation range of the tool magazine, control the tool magazine to return to zero alarm;

Handling: Press [Reset] to cancel the alarm, and manually control the tool magazine return condition to zero at the position of the tool magazine back to the origin point in the zero return mode.

A013.7 Execute the T command knife number before changing the tool

Alarm reason: After opening the tool magazine function, the T code alarm is not specified before executing the tool change command M6;

Handling: Press [Reset] to cancel the alarm and specify the T code before executing the tool change command M6.

A014.0 Z-axis does not return to mechanical zero point, and the tool cannot be changed

Alarm reason: After turning on the tool magazine function, the tool change is performed when the Z axis does not return to zero;

Handling: Press [Reset] to cancel the alarm and specify the T code before executing the tool change command M6.

A014.1 (Bucket type) Cutterhead is not in the counting position and cannot be entered

The cutterhead (disc type) is not in the counting position and cannot be reversed

The cutterhead (**turret type) is not in the counting position and the Z-axis cannot be moved**

Alarm reason: When the cutterhead of the bucket type tool magazine is not in the counting position, control the tool magazine forward alarm;

When the disc type tool magazine cutterhead is not in the counting position, control the tool sleeve to reverse the tool alarm;

When the turret-type tool magazine cutterhead is not in the counting position, control the Z-axis movement alarm;

Handling: Press [Reset] to cancel the alarm, and the manual control tool magazine cutterhead must be in the counting position.

A014.2 (bucket type, disc type) The tool magazine is not at the origin position and the spindle is not rotatable

The (turret type) Z-axis is in the tool change interval, and the spindle is not rotatable

Alarm reason: When the bucket tool magazine is not in the back position, the spindle is activated to generate an alarm;

When the disc type tool changer ATC tool arm is not at the origin, the spindle is activated to generate an alarm;

Turret tool magazine: when the Z axis is in the tool change interval, the spindle is activated to generate an alarm;

Handling: Press [Reset] to cancel the alarm and manually control the tool magazine to return to the origin position.

A014.3 Spindle tool loosening, tool change cannot be performed

Alarm reason: After opening the tool magazine function, the tool change alarm is performed when the tool is released;

Handling: Press [Reset] to cancel the alarm, and change the tool after the tool is clamped.

A014.4 The current cutterhead number has a knife and cannot be entered

Alarm reason: After opening the tool magazine function, the tool magazine forward alarm when the current

cutterhead has a tool;

Handling: Press [Reset] to cancel the alarm, the tool magazine returns to zero or reset the spindle tool number D105.

A014.5 The current cutterhead number and spindle have knives, and cannot be entered

Alarm reason: After opening the tool magazine function, the current cutterhead and spindle are detected to control the tool magazine to enter the tool alarm;

Handling: Press [Reset] to cancel the alarm and remove the knife on the spindle.

A014.6 The current spindle tool number is zero, the spindle has a knife, and the tool cannot be changed

Alarm reason: After opening the tool magazine function, when the current spindle tool number is 0, the spindle detects that the knife has a knife;

Handling: Press [Reset] to cancel the alarm and remove the knife on the spindle.

A014.7 The spindle has a knife, but the Z axis is not in the knife loading position, so the knife cannot be entered

Alarm reason: After turning on the tool magazine function, the current spindle detects a knife, and the Z axis is not at the second reference point, control the tool magazine to enter the tool alarm;

Processing: Press [Reset] to cancel the alarm and move the Z axis to the second reference point.

A015.0 Tool magazine tool number counting signal detection is abnormal, tool magazine needs to return to zero

Alarm reason: After turning on the tool magazine function, the tool magazine rotation is stopped and the tool magazine counting switch is not detected after the T102 set time has passed;

Handling: Press [Reset] to cancel the alarm and check whether the T102 setting time is correct (T102=2000); Check the tool changer counter for damage; Whether the tool changer counter level is selected correctly (K30.5).

A015.1 The origin of the tool changer is missing and the tool changer needs to be returned to zero

Alarm reason: After turning on the tool magazine function, when there is a tool magazine zero switch: the tool magazine zero switch is not detected when the cutterhead count is 1 or the cutterhead count is not 1 when the tool magazine zero switch is detected;

Handling: Press [Reset button] in zero return mode to cancel the alarm, and the tool magazine returns to zero.

A015.2 Tool changer zero return detection abnormality

Alarm reason: After turning on the tool magazine function, the tool magazine returned to zero for more than 5 minutes without returning to zero to complete the alarm;

Handling: Press the [Reset button] to cancel the alarm, and the tool magazine returns to zero.

A015.3 Tool magazine zero position set successfully

Alarm reason: After turning on the tool magazine function, the tool magazine returns to zero successfully or holds down the [machine tool zero point button] for 7 seconds after alarm;

Handling: Press [Reset button] to cancel the alarm, and the tool magazine count has been redesigned.

A015.4 Abnormal tool change stop to ensure that the tool changer is normal

Alarm reason: After turning on the tool magazine function, this alarm is issued when there is an emergency stop, reset, or alarm during the tool change process;

Handling: Press the [Reset button] to cancel the alarm and check whether the tool magazine is normal

A015.5 No command tool numbers or duplicate command tool numbers in the tool table

Alarm reason: After turning on the tool magazine function, the tool magazine command exceeds the tool magazine capacity or there are repeated values in the tool magazine data table D0-D99 when the tool magazine is searching;

Handling: Press [Reset button] to cancel the alarm, command the correct T command, or (when returning the knife) modify the current knife number D105 of the spindle; Check tool magazine datasheets D0-D99 for errors.

A015.6 The tool magazine is in the forward state, and the spindle tool cannot be moved without loosening the Z axis

Alarm reason: After turning on the tool magazine function, when the tool magazine is in the forward state, when the tool is not released, move the Z axis alarm;

Handling: Press [Reset button] to cancel the alarm, when the tool magazine is in the forward state, the tool must be released to move the Z axis.

A015.7 (bucket type) The spindle is not positioned and the tool magazine cannot be advanced

The (turret type) spindle is not positioned and the **Z-axis cannot enter the tool change interval**

Alarm reason: (bucket type) After opening the tool magazine function, when the spindle has a positioning function, the forward alarm of the tool magazine is not positioned to control the toolchange;

(turret type) After opening the tool magazine function, when the spindle has a positioning function, the Z axis is not positioned to enter the tool change interval alarm;

Handling: Press the [Reset button] to cancel the alarm, so that the spindle is positioned, and then control the tool magazine to advance, and change K21.1 to 1 when the spindle has no positioning function.

A016.0 Tool changer forward/reverse tool in place detection abnormalities

Alarm reason: After turning on the tool magazine function, the tool magazine forward/reverse tool does not detect the forward in place signal after the T104 set time;

Handling: Press [Reset button] to cancel the alarm, check whether the T104 setting is correct (T104=3000), check whether the tool magazine is normal, and manually control the tool magazine retract/return tool.

A016.1 Tool changer back/back tool in place detection abnormality

Alarm reason: After turning on the tool magazine function, the tool magazine back/back tool does not detect the back signal after the T105 set time;

Handling: Press [Reset button] to cancel the alarm, check whether the T105 settings are correct (T105=5000), check whether the tool magazine is normal, and manually control the tool magazine retract/return.

A016.2 Tool changer forward and backward/backward/**backward tool return detection abnormalities**

Alarm reason: After turning on the tool magazine function, the tool magazine forward/reverse tool in place

and backward/back tool in place signal is sensed at the same time;
Handling: Press [Reset button] to cancel the alarm, check the installation position of the tool magazine forward and backward/backward tool back tool in place switch, and check the level selection of forward and backward / reverse tool return switch (K30.3/K30.4).

A016.3 (disc) The tool is not clamped and the tool arm cannot return to the original point
Alarm reason: when the tool is not clamped, control the ATC tool arm back to the origin;
Processing: Press [Reset button] to cancel the alarm, and the knife can be clamped to perform the tool arm back to the origin;

A016.4 Spindle tool cannot be retracted without clamping tool magazine
Alarm reason: After opening the tool magazine function, the tool is not clamped to control the tool magazine back alarm;
Handling: Press the [Reset button] to cancel the alarm, and the tool magazine can be controlled to retreat only after the tool is clamped.

A016.5 Tool changer rotation abnormality forces rotation to stop
Alarm reason: After turning on the tool magazine function, the tool magazine rotates through T110, T111, T103 time without checking the tool magazine counter signal change alarm;
Handling: Press [Reset button] to cancel the alarm, check whether there is rotation, if there is rotation, check whether the T110, T111, T103 and tool magazine counter signals are normal (T110=5000; T111=3000; T103=3000）; If there is no rotation, check the tool changer circuit for problems.

A016.6 (bucket type, disc type) tool magazine is not at the origin position and cannot be started
The (turret type) Z-axis is in the tool change section and cannot be started
Alarm reason: the bucket type tool magazine is not in the back position, start the program alarm;
When the disc type tool magazine ATC tool arm is not at the origin, start the program alarm;
Turret tool magazine: when the Z axis is in the tool change interval, start the program alarm;
Handling: Press the [Reset button] to cancel the alarm and manually control the tool magazine back to the original point.

A016.7 Tool changer cannot be reset to zero without mechanical zeroing
Alarm reason: After turning on the tool magazine function, the tool magazine has a zero switch and does not perform the tool magazine zero return alarm in mechanical zero mode;
Handling: Press the [Reset button] to cancel the alarm, and perform the tool magazine reset to zero in the mechanical reset mode.

The A017.0 tool changer is not **at the origin and the Z-axis is not movable**
Alarm reason: When the bucket type tool changer is not in the back position, move the Z axis;
When the disc magazine tool arm is not at the origin position, move the Z axis;
Handling: Press [Reset button] to cancel the alarm and manually control the tool magazine back to the origin position.

A017.1 (disc type) The knife arm is not at the origin, and the knife cannot be reversed/returned

Alarm reason: When the knife arm is not at the origin position, control the tool magazine to reverse/return the tool;
Handling: Press the [Reset button] to cancel the alarm, and return the knife arm to the origin position of the knife when manual.

A017.2 (disc type) Tool **magazine reverse knife is not in place, the knife arm is not movable**
Alarm reason: When the knife sleeve is not in the inverted position, control the ATC knife arm action;
Handling: Press the [Reset button] to cancel the alarm, and the knife sleeve is in place when manual.

A017. 3 (disc type) spindle is not positioned and the knife arm is not movable
Alarm reason: when the spindle is not positioned, control the ATC knife arm action;
Handling: Press the [Reset button] to cancel the alarm, and the knife arm action can only be performed after the spindle positioning is completed.

A017.4 (disc) Z-axis is not at reference point 2 and the knife arm is not actionable
Alarm reason: When the Z axis is not at the second reference point, control the ATC knife arm action;
Handling: Press [Reset button] to cancel the alarm, the Z axis must be at the second reference point to perform the knife arm action.

A017.5 (disc type) Tool is not clamped, the knife arm is not buckleable
Alarm reason: when the knife is not clamped, control the ATC knife arm buckle knife;
Treatment: Press the [reset button] to cancel the alarm, and the knife can be clamped to execute the knife arm buckle; 。

A017.6 (disc) The tool is not loosened and the tool arm cannot be pulled down for rotation
Alarm reason: When the tool is not released, control the ATC knife arm to pull down and rotate;
Handling: Press the [Reset button] to cancel the alarm, and the knife can be released to perform the knife arm action.

A017.7 (disc) tool arm running for too long
Alarm reason: The running time of the knife arm exceeds the T115 set time alarm;
Handling: Press the [Reset button] to cancel the alarm, check whether the T115 data setting is reasonable, and check whether the brake signal of the knife arm is valid.

A018.0 (disc type) knife **arm buckle knife is not in place**
Alarm reason: When performing the knife buttoning action, the knife buckle signal alarm is not detected after the T112 set time
Handling: Press the [Reset button] to cancel the alarm, check whether the T112 data settings are reasonable, and check whether the knife buckle signal is valid.

A018.1 (disc type) The blade arm pulls down and rotates not in place
Alarm reason: When performing the pull-down rotation action, the knife snap signal alarm was not detected after the T113 set time
Handling: Press the [reset button] to cancel the alarm, check whether the T113 data settings are reasonable,

and check whether the knife buckle signal is valid.

A018.2 (disc type) The knife arm is not in place to return to the origin

Alarm reason: When performing the return to origin action, the origin point signal alarm is not detected after the T114 set time

Handling: Press the [reset button] to cancel the alarm, check whether the T114 data settings are reasonable, and check whether the origin to place signal is valid.

A018.3 (disc type) knife arm is operating abnormally

Alarm reason: When performing the knife arm action, the system issues a reset, alarm, and emergency stop to generate an alarm;

Handling: Press the [Reset button] to cancel the alarm.

A018. 4 M22 knife seeking abnormality

Alarm reason: When performing the M 22 tool seeking action, an alarm is generated when the tool magazine stops rotating after T117 time M 22 code is not completed;

Handling: Press the [Reset button] to cancel the alarm.

A020.0 Guard door open warning

Alarm reason: After opening the protective door detection function, a warning is issued when the protective door signal is not detected;

Handling: Press [Reset button] to cancel the alarm when closing the protective door, and the program cannot be started when the warning is triggered.

A020.1 Insufficient lubricant warning

Alarm reason: a warning is issued after the delay T03 set time is detected when the lubricating oil is insufficient;

Treatment: add lubricating oil and press [Reset button] to cancel the warning; Check whether the lubricating oil detection level is selected correctly (K16.2).

A020.2 Insufficient air pressure warning

Alarm reason: a warning is issued after the delay T02 set time is detected when the air pressure is insufficient;

Treatment: detect whether the intake element is normal, wait for the air pressure to rush to cancel the warning; Check whether the insufficient air pressure detection level is selected correctly (K20.2), and the program cannot be started when the warning is triggered.

A020.3 Spindle tool number zero warning

Alarm reason: not in the tool change process spindle when the front knife number is 0 to warn;

Handling: Check whether there is a knife on the spindle, remove the tool and re-perform the tool change or modify the current tool number value D105 of the spindle.

A020.4 Tool magazine debugging function opens warning

Alarm reason: The tool magazine debugging function is turned on K30.7=1 to trigger the warning;

Processing: After confirming that the tool magazine is normal, close the debugging function to cancel the warning, and the program cannot be started when the warning is triggered.

A020.5 Tool changer zero in warning

Alarm reason: trigger a warning when the tool magazine tool changer returns to zero;

Processing: The warning is canceled after the tool changer returns to zero, and the program cannot be started when the warning is triggered.

A020.6 (Bucket Pipe) Tool changer does not set back in place warning

The (disc) knife arm is not at the origin warning

(Turret type) Z-axis warns in the tool change interval

Alarm reason: (bucket type) is not in the process of tool change, the tool magazine does not trigger a warning in the back position;

(disc type) is not in the process of tool change, the knife arm does not trigger a warning at the origin position;

(Turret type) is not in the tool change process, the Z-axis triggers a warning at the position of the tool change interval

Handling: Control the tool changer to fall back to the tool magazine back position to suppress the warning, the warning does not start when triggered.

Appendix III: Tool magazine debugging

First, row knife magazine

Relevant signals

Signal	The spindle is oriented into place	Spindle directional output	Tool loosening/clamping	The knife is released into place	The tool is clamped in place
Diagnostic address	X4.2	Y4.2	Y0.2	X1.0	X1.1
Interface pins	CN15.08	CN15.22	CN62.3	CN62.09	CN62.10
remark		M19/M18	M16/M17	Normally open	Normally open

Description of the relevant parameters of the row knife

parameter	**serial number**	**illustrate**
Number	**#052**	**The 1st reference point value** of **the Z axis (set to 0 by default).**

parameters		
Number parameters	**#057**	**The 2nd** reference point value of the Z axis **(tool change downward loading point).**
Number parameters	**#062**	**Z-axis 3rd reference point value (loosening knife up point).**
Note: The reference point of Z axis 1 is set to 0 by default, the second reference point is the tool magazine tool loading position point, and the third reference point is the main axis after the main axis is loosened, and the spindle rises the tool seeking position point. (are machine coordinate values).		
bit parameters	**#23.5**	**Z-axis tool change slow interval degree limit (0: invalid 1: effective).**
bit parameters	**#23.6**	**X-axis tool change slow interval speed limit (0: invalid 1: valid).**
bit parameters	**#23.7**	**Y-axis tool change slow interval speed limit (0: invalid 1: valid).**
Number parameters	**#216**	**Z-axis tool change slow range limits speed F** to 1000 default
Number parameters	**#218**	**X-axis tool change slow interval limits speed F** default to 1000
Number parameters	**#219**	**Y-axis tool change slow range limits speed F** default to 1000
Number parameters	**#228**	**Z-axis tool change slow speed interval minimum (tool change slow interval) F23.5** Z-axis machine coordinates
Number parameters	**#229**	**The maximum value of the Z-axis tool change slow interval**
Number parameters	**#367**	**Minimum value of X-axis tool change slow interval (tool change slow interval) F23 2** X-axis machine coordinates
Number parameters	**#368**	**The maximum value of the X-axis tool change slow interval**
Number parameters	**#380**	**Y-axis tool change slow interval minimum (tool change slow interval) F23 3** Y-axis machine coordinates
Number parameters	**#381**	**The maximum value of the Y-axis tool change slow interval**
Note: If the tool changer needs to pass the slow speed at a slow speed during the tool change interval, turn on the tool change slow speed limit function #23.5=1 or #23.6=1 or #23.7=1 (depending on which axis is required for the tool magazine), and the speed is determined by the number parameter #216 or #218 or #219 set numerical control, this parameter controls the speed of G 0/G53/G28/G30 through the tool change interval; When the speed of G 0/G53/G28/G30 is lower than the speed set by parameter #216, pass through the tool change slow interval at the speed of G 0/G53/G28/G30 ； If the tool changer does not need to pass slowly in the tool change interval, do not turn on the change speed limit function #23.5=0 or #23 6=0 or #23 7=0。		
Memorize variables	**#900**	**Coordinates of the No. 1 tool machine (X-axis).**
Memorize	**#901**	**Coordinates of the No. 1 tool machine (Y/A axis).**

variables		
Memorize variables	**#902**	**Coordinates of the No. 2 tool machine (X-axis).**
Memorize variables	**#903**	**Coordinate of the No. 2 tool machine (Y/A axis).**
Memorize variables	**#904**	**Coordinates of the No. 3 tool machine (X-axis).**
Memorize variables	**#905**	**Coordinate of the No. 3 tool machine (Y/A axis).**
Memorize variables	**#906**	**Coordinates of the No. 4 tool machine (X-axis).**
Memorize variables	**#907**	**Coordinates of the No. 4 tool machine (Y/A axis).**
Memorize variables	**#908**	**Coordinates of the tool No. 5 tool (X-axis).**
Memorize variables	**#909**	**Coordinates of the No. 5 tool machine (Y/A axis).**
Memorize variables	**...**	**...**
Memorize variables	**...**	**...**
Memorize variables	**...**	**...**
PLC parameters	**K001.0**	**(0: Forbidden 1: Allow) Tool changer use** Select 1 when using the tool changer
PLC parameters	**K001.1**	**Tool magazine usage (0: other tool changers 1: row of tools)** Select 1 when using tool row tools
PLC parameters	**K001.3**	**When the tool changer function is turned on (0: use 1: do not use) the tool changer** is selected when using the tool changer
PLC parameters	**K014.6**	**(0: Check 1: Do not check) Tool clamping/loosening in place signal** Select 1 when no signal is received
PLC parameters	**K014.0**	**The spindle release signal signal is (0: normally open 1: normally closed) is valid** and normally open is selected for 0
PLC parameters	**K014.1**	**The spindle clamping signal is (0: normally open 1: normally closed) is valid** and 0 is selected for normally open
PLC parameters	**K021.1**	**The spindle positioning completion detection signal is (0:normally open 1:normally closed) is valid**
PLC parameters	**K031. 0**	**When there is no tool after tool clamping (0:Yes 1:No), the clamping signal can be detected**
This parameter applies to spindles that do not detect the tool clamping signal when the spindle is not loaded, and this parameter can be set to 1 when this spindle is such a spindle.		
PLC parameters	**K032.4**	**During tool change (0:No 1:Yes) shield soft stops**

In the case that the tool magazine is on one side of the working stroke, the soft limit can be set in front of the tool magazine, and the soft limit can be shielded during the tool change process through this parameter to make the tool change normal; Alarm when machining or manual/handwheel moving shaft enters soft limit.		
PLC parameters	**K032.5**	**A warning is prompted when** the spindle has **a front** tool number of 0 **(0**:**Yes 1:No).**
This parameter only prompts the spindle that the current tool number is 0, and warns not to install the knife on the spindle.		
PLC parameters	**T007**	**Tool release completion delay ms** appropriate delay (default 30ms).
PLC parameters	**T008**	**Tool clamping completion delay ms** can be appropriately delayed (default 30ms).
PLC parameters	**T009**	**Spindle tool release detection delay ms** (default 5000ms).
PLC parameters	**T020**	**Spindle tool clamping detection delay** ms (default 5000ms).
PLC parameters	**T004**	**Spindle orientation completion delay** ms is appropriately delayed (default 30ms).
PLC parameters	**T018**	**Spindle orientation into place signal delay detection** ms (default 2000ms).
PLC parameters	**T023**	**Spindle orientation in place detection delay** ms (default 10000ms).
PLC parameters	**C100**	**Total number of tool positions** C100 must = total number of tool magazine tool positions
PLC parameters	**D100**	**The total number of tool positions** D100=C100 is not entered
PLC parameters	**D101**	**Command knife number** The programming input is not modifiable
PLC parameters	**D102**	**The previous knife number of the command knife number** is not automatically calculated
PLC parameters	**D103**	**Cutterhead counting position** The automatic calculation is not modifiable
PLC parameters	**D104**	**Manual tool changer tool number** The automatic calculation cannot be modified
PLC parameters	**D105**	**Spindle current tool number** The current spindle tool number can be manually modified when it is unreasonable
PLC parameters	**D120**	**Tool magazine return / tool search address** Automatic calculation cannot be modified
PLC parameters	**D121**	**Tool changer return / tool search output results** Automatic calculations cannot be modified
PLC parameters	**D124**	**Tool magazine data table search range** The automatic calculation cannot be modified D124=D100+1

PLC parameters	**D126**	**Spindle zero speed output range (r/min)** D126==30
PLC parameters	**D000**	**Cutterhead No. 0 Pocket Setting (Tool Change Number Retrieval Table Address)** D000==0
PLC parameters	**D001**	**Cutterhead No. 1 socket set** D001==1
PLC parameters	**D002**	**Cutterhead No. 2 socket setting** D002==2
PLC parameters	**D003**	**Cutterhead No. 3 socket set** D003==3
PLC parameters	**D004**	**Cutterhead No. 4 socket setting** D004==4
...	**...**	**...**
...	**...**	**...**
...	**...**	**...**
PLC parameters	**D097**	**Cutterhead 97 pocket setting** D097==97
PLC parameters	**D098**	**Cutterhead 98 pocket setting** D099==98
PLC parameters	**D099**	**Cutterhead 99 pocket setting** D099==99

Concentrate:

Row knife change macro program

Note: If the tool changer does not need to be positioned, delete the M19/M18 in the program

Choose different tool change programs according to different tool magazines

Then select the corresponding macro variable according to the tool changer

In the system, O 9105/O9106/O9108/O9109/O9110 correspond to this type of row of tools, and different options are made according to different tool changing actions

Change the tool program, and then change the program name to O9101;

#900 No. 1 knife X position	#920 11 knife X position	#940 No. 21 knife X position	#961 No. 12 knife Z position
#901 No. 1 knife Y/A position	#921 No. 11 knife Y/A position	#941 21 gauge knife Y/A position	#962 13 knife Z position
#902 No. 2 knife X position	#922 No. 12 knife X position	#942 No. 22 knife X position	#963 14 knife Z position
#903 No. 2 knife Y/A position	#923 12 size knife Y/A position	#943 22 knife Y/A position	#964 15 knife Z position
#904 No. 3 knife X position	#924 13 knife X position	#944 No. 23 knife X position	#965 16 knife Z position
#905 No. 3 knife Y/A position	#925 13 size knife Y/A position	#945 23 gauge knife Y/A position	#966 17 knife Z position

#906 No. 4 knife X position	#926 14 knife X position	#946 No. 24 knife X position	#967 18 knife Z position
#907 No. 4 knife Y/A position	#927 14 gauge knife Y/A position	#947 24 knife Y/A position	#968 19 knife Z position
#908 No. 5 knife X position	#928 15 knife X position		#969 20 knife Z position
#909 No. 5 knife Y/A position	#929 15 knife Y/A position	#950 No. 1 knife Z position	#970 21st knife Z position
#910 No. 6 knife X position	#930 16 knife X position	#951 No. 2 knife Z position	#971 22 knife Z position
#911 No. 6 knife Y/A position	#931 16 size knife Y/A position	#952 No. 3 knife Z position	#972 23rd knife Z position
#912 No. 7 knife X position	#932 17 knife X position	#953 No. 4 knife Z position	#973 24 knife Z position
#913 No. 7 knife Y/A position	#933 17 knife Y/A position	#954 No. 5 knife Z position	
#914 No. 8 knife X position	#934 18 knife X position	#955 No. 6 knife Z position	
#915 No. 8 knife Y/A position	#935 18 gauge knife Y/A position	#956 No. 7 knife Z position	
#916 No. 9 knife X position	#936 19 knife X position	#957 No. 8 knife Z position	
#917 9 size knife Y/A position	#937 19 knife Y/A position	#958 No. 9 knife Z position	
#918 10 knife X position	#938 20 number knife X position	#959 10 knife Z position	
#919 No. 10 knife Y/A position	#939 20 gauge knife Y/A position	#960 11 knife Z position	

2. Bucket type tool magazine

Precautions for the application of the bucket type tool changer

1. During the tool change process, it must be ensured that each step is completed before the next action can be carried out;
2. Do not enter the tool until the tool magazine positioning sensor is confirmed
3. The spindle has not changed the tool point, and the tool cannot be changed;
4. The spindle positioning is not completed, and the tool cannot be changed;
5. Before the tool magazine is returned, the spindle cannot be rotated and the Z-axis cannot be moved;
6. When the voltage of the tool magazine motor is too high, the overload protection relay should trip

off first to avoid burning out the tool magazine motor;

7. Please change the tool manually first to check that the tool magazine forward, backward, tool magazine forward and reverse, spindle grab loose knife and other actions are correctly coordinated, and then start CNC automatic sequence program control;
8. During the debugging process, only the input and output signals (X/Y address) of the ladder diagram need to be modified to complete the debugging requirements; If you are not sure, do not modify the others to avoid accidents.

Relevant signals

Foot size	address	definition	remark
CN61.03	X0.2	Manual loose/clamp control	Normally open contacts
CN61.09	X1.0	The knife is released into place signal	Normally open contacts
CN61.10	X1.1	Tool clamping in place signal	Normally open contacts
CN61.11	X1.2	Tool changer return to zero to place signal	Level modification K30.2
CN61.29	X2.0	Tool changer forward in place signal	Level modification K30.3
CN61.30	X2.1	Tool changer back into place signal	Level modification K30.4
CN61.31	X2.2	Tool changer counting switching signals	Level modification K30.5
CN61.35	X2.6	The current cutterhead has a tool detection signal	Level modification K32.3
CN61.36	X2.7	The spindle has a knife detection signal	Level modification K32.1
CN15.08	X4.2	The spindle is directed to the signal in place	Level modification K21.1
CN61.03	Y0.2	Tool loosening/clamping output	M16/M17
CN61.38	Y3.1	Tool changer forward output	M21/M22
CN61.39	Y3.2	The tool changer reverses the output	M21/M22
CN61.40	Y3.3	Tool changer forward output	M21
CN61.41	Y3.4	The tool changer outputs backwards	M24

CN61.42	Y3.5	Tool changer back to position indicator	
CN15.22	Y4.2	Spindle directional output	M19/M18

Note: M21/M22/M24 cannot be used alone.

parameter	**serial number**	**illustrate**
Number parameters	**#052**	**The 1st reference point value** of **the Z axis (set to 0 by default).**
Number parameters	**#057**	**The 2nd** reference point value of the Z axis **(tool change downward loading point).**
Number parameters	**#062**	**Z-axis 3rd reference point value (loosening knife up point).**
Note: The reference point of Z axis 1 is set to 0 by default, the second reference point is the tool magazine tool loading position point, and the third reference point is the main axis after the main axis is loosened, and the spindle rises the tool seeking position point. (All are machine coordinate values)		
bit parameters	**#23.5**	**Z-axis tool change slow interval degree limit (0: invalid 1: effective).**
bit parameters	**#23.6**	**X-axis tool change slow interval speed limit (0: invalid 1: valid).**
bit parameters	**#23.7**	**Y-axis tool change slow interval speed limit (0: invalid 1: valid).**
Number parameters	**#216**	**Z-axis tool change slow range limits speed F** to 1000 default
Number parameters	**#218**	**X-axis tool change slow interval limits speed F** default to 1000
Number parameters	**#219**	**Y-axis tool change slow range limits speed F** default to 1000
Number parameters	**#228**	**Z-axis tool change slow speed interval minimum (tool change slow interval) F23.5** Z-axis machine coordinates
Number parameters	**#229**	**The maximum value of the Z-axis tool change slow interval**
Number parameters	**#367**	**Minimum value of X-axis tool change slow interval (tool change slow interval) F23 2** X-axis machine coordinates
Number parameters	**#368**	**The maximum value of the X-axis tool change slow interval**
Number parameters	**#380**	**Y-axis tool change slow interval minimum (tool change slow interval) F23 2** Y-axis machine coordinates
Number parameters	**#381**	**The maximum value of the Y-axis tool change slow interval**
Note: If the tool changer needs to pass the slow speed at a slow speed during the tool change interval, turn on the tool change slow speed limit function #23.5=1 or #23.6=1 or #23.7=1		

(depending on which axis is required for the tool magazine), and the speed is determined by the number parameter #216 or #218 or #219 set numerical control, this parameter controls the speed of G 0/G53/G28/G30 through the tool change interval; When the speed of G 0/G53/G28/G30 is lower than the speed set by parameter #216, pass through the tool change slow interval at the speed of G 0/G53/G28/G30 ； If the tool changer does not need to pass slowly in the tool change interval, do not turn on the change speed limit function #23.5=0 or #23 6=0 or #23 7=0。		
PLC parameters	**K014.6**	**(0: Check 1: Do not check) Tool clamping/loosening in place signal** Select 1 when no signal is received
PLC parameters	**K014.0**	**The spindle release signal is (0: normally open 1: normally closed) is valid** and 0 is selected for normally open
PLC parameters	**K014.1**	**The spindle clamping signal is (0: normally open 1: normally closed) is valid** and 0 is selected for normally open
PLC parameters	**K021.1**	**The spindle positioning completion detection signal is (0:normally open 1:normally closed) is valid**
PLC parameters	**K030.1**	**Tool changer (0:No 1:Yes) with a return switch** Select 1 when there is a zero switch
PLC parameters	**K030.2**	**The tool changer zero switch signal is (0:NO 1:NC) active**
PLC parameters	**K030.3**	**The tool magazine forward/reverse tool in place signal is (0:normally open 1:normally closed) is valid**
PLC parameters	**K030.4**	**The tool magazine back/return signal is (0:normally open 1:normally closed) is valid**
PLC parameters	**K030.5**	**The tool magazine count switch signal is (0:normally open 1:normally closed) active**
PLC parameters	**K030.7**	**The tool changer (0:No 1:Yes) enters debug mode**
PLC parameters	**K031. 0**	**When there is no tool after tool clamping (0:Yes 1:No), the clamping signal can be detected**
This parameter applies to spindles that do not detect the tool clamping signal when the spindle is not loaded, and this parameter can be set to 1 when this spindle is such a spindle.		
PLC parameters	**K031.2**	**Output after the tool changer is forward/reversed in place (0: uninterrupted 1: disconnected).**
PLC parameters	**K031.3**	**Output after the tool magazine is back/back in place (0: uninterrupted 1: disconnected).**
PLC parameters	**K032.0**	**Spindle with knife detection (0: invalid 1: valid)**
See whether the machine tool has a tool detection signal for the spindle installed, generally not installed, select 0		
PLC parameters	**K032.1**	**The spindle tool detects that the switch signal is (0:normally open 1:normally closed) active**
PLC	**K032.2**	**Current cutterhead changer with cutter detection (0: invalid 1: valid)**

parameters		
See whether the machine tool has an installed cutterhead with a tool detection signal, generally not installed, select 0		
PLC parameters	**K032.3**	**The current cutterhead detection switch signal is (0:NO 1:NC) active**
PLC parameters	**K032.4**	**During tool change (0:No 1:Yes) shield soft stops**
The tool changer does not need to set this parameter and is 0.		
PLC parameters	**K032.5**	**A warning is prompted when** the spindle has **a front** tool number of 0 **(0:Yes 1:No).**
This parameter only prompts the spindle that the current tool number is 0, and warns not to install the knife on the spindle.		
PLC parameters	**K032.6**	**The changer cutterhead rotation (0:No 1:Yes) has only forward rotation**
PLC parameters	**T007**	**Tool release completion delay** Appropriate delay (default 30ms).
PLC parameters	**T008**	**Tool clamping completion delay** Appropriate delay (default 30ms).
PLC parameters	**T009**	**Spindle tool release detection delay** T009==5000 (default 5000ms).
PLC parameters	**T020**	**Spindle tool clamping detection delay** T020==5000 (default 5000ms).
PLC parameters	**T004**	**Spindle orientation completion delay** T004==30 (default 30ms).
PLC parameters	**T018**	**Spindle orientation into place signal delay detection** T018==2000 (default 2000ms).
PLC parameters	**T023**	**Spindle orientation in place detection delay** T023==10000 (default 10000ms).
PLC parameters	**T100**	**Tool changer manual forward key delay** T100==0 (default 0ms).
PLC parameters	**T101**	**Tool changer manual reverse key delay** T101==0 (default 0ms).
PLC parameters	**T102**	**Tool changer rotation stop delay detection** T102==0 (default 0ms).
PLC parameters	**T103**	**Tool magazine counter delay detection** T103==2000 (default 2000ms).
PLC parameters	**T104**	**Tool magazine tool feed/reverse tool delay detection** T104==5000 (default 5000ms).
PLC parameters	**T105**	**Tool magazine tool retract/return delay detection T105==5000** (default 5000ms).
PLC	**T106**	**Tool magazine feed/reverse tool completion delay** T106==50 (default

parameters		50ms).
PLC parameters	**T107**	**Tool magazine retract/return completion delay** T107==50 (default 50ms).
PLC parameters	**T108**	**Tool change forward and backward detection abnormal delay** T108==0 (default 0ms).
PLC parameters	**T109**	**Tool changer zero-to-bit signal delay detection** T109==0 (default 0ms).
PLC parameters	**T110**	**Tool magazine rotation delay detection** T110==3 000 (default 3000ms).
PLC parameters	**T111**	**Tool magazine counter in place delay detection** T111==2 000 (default 2000ms).
PLC parameters	**T116**	**Tool changer counter in place delay** T116==50 (default 50ms).
PLC parameters	**T117**	**M22 Finder detection delay** T117==50000 (default 50000ms).
PLC parameters	**C100**	**Total number of tool positions** C100 must = total number of tool magazine tool positions
PLC parameters	**D100**	**The total number of tool positions** D100=C100 is not entered
PLC parameters	**D101**	**Command knife number** The programming input is not modifiable
PLC parameters	**D102**	**The previous knife number of the command knife number** is not automatically calculated
PLC parameters	**D103**	**Cutterhead counting position** The automatic calculation is not modifiable
PLC parameters	**D104**	**Manual tool changer tool number** The automatic calculation cannot be modified
PLC parameters	**D105**	**Spindle current tool number** The current spindle tool number can be manually modified when it is unreasonable
PLC parameters	**D120**	**Tool magazine return / tool search address** Automatic calculation cannot be modified
PLC parameters	**D121**	**Tool changer return / tool search output results** Automatic calculations cannot be modified
PLC parameters	**D124**	**Tool magazine data table search range** The automatic calculation cannot be modified D124=D100+1
PLC parameters	**D126**	**Spindle zero speed output range (r/min)** D126==30
PLC parameters	**D000**	**Cutterhead No. 0 Pocket Setting (Tool Change Number Retrieval Table Address)** D000==0
PLC parameters	**D001**	**Cutterhead No. 1 socket set** D001==1
PLC parameters	**D002**	**Cutterhead No. 2 socket setting** D002==2

PLC parameters	**D003**	**Cutterhead No. 3 socket set** D003==3
PLC parameters	**D004**	**Cutterhead No. 4 socket setting** D004==4
...	**...**	**...**
...	**...**	**...**
...	**...**	**...**
PLC parameters	**D097**	**Cutterhead 97 pocket setting** D097==97
PLC parameters	**D098**	**Cutterhead 98 pocket setting** D099==98
PLC parameters	**D099**	**Cutterhead 99 pocket setting** D099==99

Note: C100 and D100 must be less than 100, and the total capacity of the tool magazine is 99; When C100=16, the D000-D016 data is valid; When C100=24, the D000-D024 data is valid.

Relevant M code

M16	The knife is released
M17	Tool clamping
M18	Untarget
M19	Spindle orientation
M21	Tool magazine forward
M22	Tool magazine finder
M24	The tool magazine is set back
M50	The knife change begins
M51	End of knife change

Tool changer installation and related parameter settings

This ladder diagram requires the tool changer machine to be as follows:

1. The spindle has a tool loosening/clamping device, and the loosening/clamping device can be selected with or without a loosening/clamping in place signal
2. The spindle can choose whether there is a positioning function, if there is no positioning function, the M19/M18 in the tool changer O9101 needs to be deleted
3. Tool magazine capacity is less than 100
4. The tool magazine has forward and reverse changes
5. The tool magazine has counting switch, forward in place detection switch, and back in place detection switch
6. The tool changer can be selected with or without a zero switch

If the tool changer does not have a zero switch, and each detection switch of the tool magazine is normally open, the K parameter is set

K030.1==0; K030.3==0; K030.4==0; K030.5==0;

Bucket type tool magazine control conditions:

1. Automatic tool changer procedure steps and conditions:

1) Perform the tool change procedure M06 Txxx; To determine whether to change the tool, the machine tool lock, auxiliary lock, and spindle knife number are equal to the T command knife number, and the machine tool does not change the tool;
2) Tool change starts, spindle, cooling, modal values are canceled;
3) Automatic tool change starts; Determine whether the automatic tool change conditions are met, if they are not met, the alarm, automatic tool change conditions: the correct T command has been executed, the Z axis has been returned to zero to complete, and the tool is in the clamping state;
4) Determine whether to skip the return part, the spindle knife number is 0 or the spindle detects that there is no knife;
5) The tool magazine returns the knife, the tool magazine rotates to find the knife number to be returned, and the tool magazine advances after finding the knife number and sensing the counter;
6) spindle loosening knife;
7) Determine whether to skip the tool hunting part, and jump when the command knife number Txxx is T0;
8) Tool magazine to find the tool, the tool magazine rotates to find the command T tool number, and the tool magazine advances after finding the tool number and sensing the counter;
9) spindle clamping knife;
10) The tool magazine is retreated, and the spindle can only be retreated after the knife is clamped;

2. Manual control of tool magazine conditions:

1) The manual rotation or forward and backward of the tool magazine must be effective in the manual/handwheel mode, and the tool magazine zero operation must be effective in the mechanical zero mode;
2) The tool magazine controls the forward conditions, the system is in standby state, in the manual/handwheel mode, the spindle tool number is consistent with the current cutterhead number of the tool magazine, the Z axis is on the second reference point, the tool magazine cutterhead counter is sensed, and the spindle positioning is completed;
3) Tool magazine control back condition, the system in standby state, in manual/handwheel mode, tool clamping state;
4) The tool changer controls the rotation conditions, the system is in standby, in manual/handwheel mode, the tool magazine is set back in place or the Z axis is in the 3rd reference point position;
5) Tool magazine zero control condition, tool magazine is equipped with a zero switch, the system is in standby state, in the mechanical zero mode, the tool magazine is in the back position.

Tool magazine debugging:

Notes:

1. Confirm whether the system emergency stop is valid
2. Confirm that the tool changer line connection is normal
3. Confirm that the tool changer has no foreign objects in its range of movement
4. There is a special tool magazine control interface in the ladder diagram interface, and the interface can be entered through the F function key corresponding to [tool magazine back zero key], [tool

magazine forward rotation key], [tool magazine reversal key], [forward/reverse tool key], [back/back tool key], [tool arm control key], [Z axis back to the second reference point key], [Z axis back to the third reference point key];

To enter the control interface, press [ladder diagram] → [tool magazine control].

Tool changer manual and zero operation:

Manually control the tool magazine action to detect whether each step of the tool magazine is normal.

1. The tool changer usage function works
   A. The system default Ladder01.grp ladder diagram is the bucket tool magazine ladder diagram, with a number parameter of 304==1 (requires restart);
   B. PLC parameter K1.0==1; K1.1==0; K1.3=0;

2. Manually confirm the direction of tool magazine rotation
   A. Modify K30.7==1 to open the tool magazine debugging function and manually control the tool magazine condition removal
   B. In the manual/handwheel mode, press [tool magazine forward rotation key] tool magazine to rotate in the direction of increasing the tool number, press [tool magazine reversal key] tool magazine to rotate in the direction of decreasing the tool number, if the rotation direction is opposite, the tool magazine count will be disordered resulting in tool exchange errors, and adjusting the phase sequence of the tool magazine motor can be solved.

3. Manually confirm whether the tool changer forward and backward is normal
   A. Modify K30.7==1 to open the tool magazine debugging function and manually control the tool magazine condition removal
   B. The Z-axis return to zero ensures that the tool changer is pushed out without colliding with the spindle
   C. Press the [Tool Magazine Forward Key] tool magazine in the manual/handwheel mode, press the [Tool Magazine Back Button] tool magazine to retract, if the direction is opposite, you can modify the solenoid valve or change the air change pipe to solve.

4. Tool magazine zero operation
   A. If the tool changer has a zero switch, modify K30.1==1;
      1) Manually return the tool magazine tool back position, and then switch to the mechanical zero return mode and press the [tool magazine return zero button] The tool magazine is turning back to zero, and at the same time issue an A20.5 tool magazine zero return warning;
      2) When the tool magazine zero return is completed, the warning in the tool magazine return to zero is canceled, and the A15.3 tool magazine origin is set successfully, press [Reset button] to cancel the alarm, the tool magazine zero return is completed, and the spindle tool number becomes 0 after the zero return is completed.
      3) If the tool changer zero return signal is not detected after more than 5 minutes of zeroing, the system alarms A15.2 tool changer zero return detection abnormality.

   B. If the tool changer does not have a zero switch, modify K30.1==0
      1) Press [Tool magazine forward rotation key] or [tool magazine reversal key] in manual/handwheel

mode to turn the tool magazine cutterhead to the No. 1 cutterhead position;

2) Press and hold the [Machine Tool Zero Point Key] for 7 seconds, the system sends out that the A15.3 tool magazine origin is set successfully, press the [Reset button] to cancel the alarm, and the tool magazine origin setting is completed.

5. Spindle positioning angle and tool change coordinate adjustment

A. Spindle positioning angle adjustment (refer to spindle servo drive manual).

Spindle positioning angle adjustment steps (examples):

1) The system enters the operation level above level 2, open the tool magazine debugging function K30.7==1 The system issues an A20.4 tool magazine debugging function opening warning, and the tool magazine can be unconditionally entered and retreated;
2) Perform Z-axis zeroing in the mechanical zero point mode, wait for the Z-axis to be completed to zero, if the height of Z-axis zeroing is not high, you can raise the Z axis manually/handwheel handle to avoid the tool magazine forward colliding with the spindle;
3) Prepare a handle and remove the handle nails;
4) The system switches to manual or handwheel mode, press [Tool magazine forward button] to make the tool magazine in the forward position of the tool magazine;
5) Install the nailless shank on the current cutterhead of the tool magazine and adjust the position;
6) The system switches to the handwheel mode, which can move the Z axis up and down (please operate with caution); At this time, directly twist the spindle, rotate for more than one week, adjust the spindle position by hand, so that the key on the spindle can enter the slot of the tool holder;
7) Record the current spindle drive display angle and set it to the drive positioning angle (refer to the drive manual);
8) The handwheel moves the Z axis up from the shank position and removes the tool magazine shank;

B. Z-axis tool change point coordinate setting

1) After adjusting the spindle positioning according to the above operation, move the Z axis to a safe position in the manual/handwheel mode, press the [spindle positioning key] to perform the spindle positioning, and then press the [tool change key] to loosen the knife, install the tool holder with a pull nail on the current cutterhead of the tool magazine to adjust the position (at this time, the tool magazine is still in the forward state and forward in place), the system switches to the handwheel mode, you can move the Z axis up and down to adjust the upward height of the Z axis of the loose knife (note that the key of the spindle in this position must be higher than the height of the tool holder tool nail in the tool magazine), record the Z-axis machine coordinates of this position, Enter the Z-axis machine coordinates into the 3rd reference point of the Z-axis (the upward point of the loose knife) [parameter number parameter #062] (generally the zero point of the Z-axis of the bucket is higher than the position of the tool magazine tool nail pulling point, when the position of the Z axis return to zero point is higher than the tool nail, the third reference point of the Z axis is set to 0, which is consistent with the 1st reference point); After setting the upward point of the loosening knife, determine that the spindle tool is in the loose state, the spindle positioning is completed, the handwheel moves up the Z axis to adjust the coordinates of the tool changing and loading point (at this time, ensure that the tool is loosened), record the adjusted Z-axis machine tool coordinates, and input the recorded Z-axis machine coordinates to the second reference point of the Z-axis (the downward loading point of the tool change) [parameter

number parameter #057];

2) Press [Tool Change Key] spindle tool clamping, press [Tool magazine back key] to return the tool magazine (make sure the tool is clamped);
3) The 2/3rd reference point parameter of the Z axis must be set, otherwise the mechanical part will be damaged;
4) Turn off the tool magazine debugging function K30.7==0; Remove the knife from the spindle;

6. After passing the above two steps (point 5), you can call the tool change program O9101 in edit mode, write the Txxx command at the beginning of the program, and then switch to automatic mode to run the program in a single section to check whether the tool change action is correct (to modify the tool change program O9101, you must put the bit parameter #029.5==0, and #029.5==1 after debugging is completed to avoid mistakenly modifying the tool change program);

The modified tool change procedure is:

O9101;

Txxx; **(Added during single-segment detection program, delete this segment after the tool change action is normal)**).

G65H81P50Q#1003R1;//G54.3=1 command knife is equal to spindle knife does not perform tool change

G69G50G15G80G40;

M05;//Spindle off

M09;// Cooling off

M50;//Automatic tool change starts

G65H81P20Q#1000R1;//G54.0=1 Spindle tool number is 0

G65H81P20Q#1001R1;//G54.1=1 No knife on spindle

M19G00G91G49G30Z0;//Z downward spindle orientation (quick to reach the second reference point)

M21;//Tool magazine forward to find the return knife

N20M19;//Spindle positioning

M16;//Knife loosening

G91G49G30Z0P3;//Z axis up (fast to third reference point).

G65H81P30Q#1002R1;//G54.2 command knife is 0

M22;//Tool magazine rotation

G91G30G49Z0;//Z axis downlink (fast to second reference point)

N30M17;//Spindle clamp cutter

M24;//Tool magazine setback

M18;// Unorient the spindle

N50M51;//Automatic tool change ends

M99;

%

7. After executing point 6, delete the Txxx command added at the beginning of the tool changer O9101, and execute the automatic tool change instruction in MDI mode after automatic mode:

Txxx M06 or M06 Txxx ;

Warning: When the spindle tool number is 0, the spindle is forbidden to load the knife, otherwise it will malfunction during automatic tool change, resulting in mechanical damage!

The correct way to deal with emergency stop, reset, alarm, and power failure encountered during the operation of the tool magazine:

1. In the process of automatic tool change, the tool magazine search for the command tool number is completed, the spindle has dropped to the tool loading position, and the tool has been clamped, but the tool magazine has not returned, in case of emergency stop, reset, alarm, power failure, if you need to continue to use the machine tool, you must press the [tool magazine back button] in the manual/handwheel mode when the tool magazine is returned, and then used; (Note: At this time, you cannot release the tool to move up the Z axis and then return to the tool magazine, so that the cutterhead corresponding to the spindle is equipped with a tool, leaving a hidden danger of hitting the knife for the next tool change);

2. In the process of automatic tool change, the tool magazine advances into place to the spindle to descend to the tool loading position, and the tool has been released, the Z axis has not been lifted or lifted, the cutterhead has not yet rotated, such as emergency stop, reset, alarm, power failure, if you need to continue to use the machine tool, you must observe whether the tool has been loosened in the manual/handwheel mode, if you do not release press the [tool change key] to release the tool, and then use the manual/handwheel method to lift the Z axis to a safe height, press the [tool change key] to clamp the tool, press the [tool magazine back button] to return to the tool magazine, (Note: The current spindle corresponding to the tool magazine cutterhead is equipped with tools, there is no knife on the spindle, and the system displays T000) can be used normally;

3. In the process of automatic tool change, when the tool magazine cutterhead rotates and selects the tool, in case of emergency stop, reset, alarm, power failure, there may be inconsistencies between the current tool number and the actual tool number, the system will prompt A015.0 (tool magazine tool number counting signal detection is abnormal, tool magazine needs to return to zero) or A015.4 (abnormal tool change stop to ensure that the tool magazine is normal) alarm, if you need to continue to use the machine tool, if the tool magazine is in the forward state, press [tool change key] in the manual/handwheel mode to release the tool, lift the Z axis to a safe height press [Tool change key] clamp the tool, press [Tool magazine back button] to return to the tool magazine, and then reset the tool number in the tool magazine (tool magazine back to zero) and reset the spindle tool number D105 (remove the tool to set to 0), you can use it normally.

Bucket type tool changer macro program and commentary

O9101; (program name).
G65H81P50Q#1004R1;//G54.4=1Do not perform tool change without tool change
G65H81P50Q#1003R1; (Machine tool lock, auxiliary lock, spindle tool number = T command tool number when jumping N50 tool change end)
G69G50G15G80G40; (Cancellation of relevant modal values).

M05;//Spindle off
M09;// Cooling off
M50; (Start from customizing the tool change, and detect whether the tool change conditions are met, and alarm if it is not met)
G65H81P20Q#1000R1; (When the spindle tool number is 0, the jump knife N20 does not perform the return operation)
G65H81P20Q#1001R1; (When the spindle is opened with a knife detection, the knife jump knife N20 is not detected and the return operation is not performed)
M19G00G91G49G30Z0; (Spindle positioning, Z-axis tool change point [2nd reference point])
M21; (Return the knife - > tool magazine rotation tool finder tool magazine forward in place)
N20M19;//Spindle positioning
M16;//Knife loosening
G91G49G30Z0P3; (Z-axis lifts [3rd reference point]).
G65H81P30Q#1002R1; (When the T command is 0, the tool seeking command is not executed to jump to N30)
M22; (Tool finder-> tool magazine rotation tool finder tool magazine forward in place)
G91G30G49Z0; (Z-axis tool change point [2nd reference point]))
N30M17;//Spindle clamp cutter
M24;//Tool magazine setback
M18;// Unorient the spindle
M51;//Automatic tool change ends
N50M99;
%

Third, disc type tool magazine

Disc tool changer application considerations

1. During the tool change process, it must be ensured that each step is completed before the next action can be carried out;
2. Do not reverse the tool until the tool magazine positioning sensor is confirmed
3. When the tool magazine is not in the tool return position, the tool magazine motor cannot be started to rotate the cutterhead;
4. Do not change the knife before the reverse knife is confirmed;
5. The spindle has not changed the tool point, and the tool cannot be changed;
6. The spindle positioning is not completed, and the tool cannot be changed;
7. Before the ATC knife arm returns to the original point, the spindle cannot be rotated and the Z axis cannot be moved;
8. When the voltage of the tool magazine motor is too high, the overload protection relay should trip off first to avoid burning out the tool magazine motor;
9. Please change the knife manually first to check that the actions such as inverting the knife, changing the tool mechanical transfer knife, spindle grabbing and loosening knife are correct, and

then start the CNC automatic sequence program control;

10. During the debugging process, only the input and output signals (X/Y address) of the ladder diagram need to be modified to complete the debugging requirements; If you are not sure, do not modify the others to avoid accidents.

Relevant signals

Foot size	address	definition	remark
CN61.03	X0.2	Manual loose/clamp control	Normally open contacts
CN61.09	X1.0	The knife is released into place signal	Normally open contacts
CN61.10	X1.1	Tool clamping in place signal	Normally open contacts
CN61.11	X1.2	Tool changer return to zero to place signal	Level modification K30.2
CN61.29	X2.0	Tool magazine reverse in place signal	Level modification K30.3
CN61.30	X2.1	Tool changer return to place signal	Level modification K30.4
CN61.31	X2.2	Tool changer counting switching signals	Level modification K30.5
CN61.34	X2.5	ATC knife arm brake signal	Level modification K31.5
CN61.35	X2.6	ATC knife arm gripping signal	Level modification K31.6
CN61.36	X2.7	ATC knife arm origin signal	Level modification K31.7
CN15.08	X4.2	The spindle directs the completion signal	Level modification K21.1
CN61.03	Y0.2	Tool loosening/clamping output	M16/M17
CN61.38	Y3.1	Tool changer forward output	M22
CN61.39	Y3.2	The tool changer reverses the output	M22
CN61.40	Y3.3	Tool magazine reverse tool output	M21
CN61.41	Y3.4	Tool changer return tool output	M24
CN61.42	Y3.5	ATC tool arm output	M23/M25/M36

CN15.22	Y4.2	Spindle directional output	M19/M18

Note: M21/M22/M23/M24/M25/M26 cannot be used alone.

parameter	serial number	illustrate
Number parameters	**#052**	**Z-axis 1st reference point value**
Number parameters	**#057**	**The 2nd** reference point value of the Z **axis (tool change downward point loading point).**
Number parameters	**#062**	**Z-axis 3rd reference point value**
PLC parameters	**K001.0**	**(0: Forbidden 1: Allow) Tool changer use** Select 1 when using the tool changer
PLC parameters	**K001.1**	**Tool magazine usage (0: other tool changers 1: row of tools)** 0 is selected when the tool row is not in use
PLC parameters	**K001.3**	**When the tool changer function is turned on (0: use 1: do not use) the tool changer** is selected when using the tool changer
PLC parameters	**K014.6**	**(0: Check 1: Do not check) Tool clamping/loosening in place signal**
PLC parameters	**K014.0**	**The spindle release signal is (0: normally open 1: normally closed) is valid** and 0 is selected for normally open
PLC parameters	**K014.1**	**The spindle clamping signal is (0: normally open 1: normally closed) is valid** and 0 is selected for normally open
PLC parameters	**K021.1**	**The spindle positioning completion detection signal is (0:normally open 1:normally closed) is valid**
PLC parameters	**K030.1**	**Tool changer (0:No 1:Yes) with a return switch** Select 1 when there is a zero switch
PLC parameters	**K030.2**	**The tool changer zero switch signal is (0:NO 1:NC) active**
PLC parameters	**K030.3**	**The tool magazine forward/reverse tool in place signal is (0:normally open 1:normally closed) is valid**
PLC parameters	**K030.4**	**The tool magazine back/return signal is (0:normally open 1:normally closed) is valid**
PLC parameters	**K030.5**	**The tool magazine count switch signal is (0:normally open 1:normally closed) active**
PLC parameters	**K030.6**	**The disc ATC arm (0:No 1:Yes) enters debug mode**
Note: When opening the knife arm to debug K30.6=1, the knife arm rotation is not limited by conditions, and the operation is prudent;		
PLC parameters	**K030.7**	**The tool changer (0:No 1:Yes) enters debug mode**
PLC parameters	**K031. 0**	**When there is no tool after tool clamping (0:Yes 1:No), the clamping signal can be detected**

PLC parameters	**K031.2**	**Output after the tool changer is forward/reversed in place (0: uninterrupted 1: disconnected).**
PLC parameters	**K031.3**	**Output after the tool magazine is back/back in place (0: uninterrupted 1: disconnected).**
PLC parameters	**K031.5**	**The disc ATC brake signal is (0:NO 1:NC) active**
PLC parameters	**K031.6**	**The disc ATC gripper signal is (0:normally open 1:normally closed) active**
PLC parameters	**K031.7**	**The disc ATC origin signal is (0:normally open 1:normally closed) valid**
PLC parameters	**K032.4**	**During tool change (0:No 1:Yes) shield soft stops**
The disc changer does not need to set this parameter and is 0.		
PLC parameters	**K032.5**	**A warning is prompted when** the spindle has **a front** tool number of 0 **(0:Yes 1:No).**
This parameter only prompts the spindle that the current tool number is 0, and warns not to install the knife on the spindle.		
PLC parameters	**K032.6**	**The changer cutterhead rotation (0:No 1:Yes) has only forward rotation**
PLC parameters	**T007**	**Tool release completion delay** can be appropriately delayed (default 30ms).
PLC parameters	**T008**	**Tool clamping completion delay** can be appropriately delayed (default 30ms).
PLC parameters	**T009**	**Spindle tool release detection delay** T009==5000 (default 5000ms).
PLC parameters	**T020**	**Spindle tool clamping detection delay** T020==5000 (default 5000ms).
PLC parameters	**T004**	**Spindle orientation completion delay** T004==3 0 (default 30ms).
PLC parameters	**T018**	**Spindle orientation in place signal delay detection** T018==2000 (default 2000ms).
PLC parameters	**T023**	**Spindle orientation in place detection delay** T023==10000 (default 10000ms).
PLC parameters	**T100**	**Tool changer manual forward key delay** T100==0 (default 0ms).
PLC parameters	**T101**	**Tool changer manual reverse key delay** T101==0 (default 0ms).
PLC parameters	**T102**	**Tool changer rotation stop delay detection** T102==0 (default 0ms).
PLC parameters	**T103**	**Tool magazine counter delay detection** T103==2000 (default 2000ms).
PLC	**T104**	**Tool magazine tool feed/reverse tool delay detection** T104==5000 (default

parameters		5000ms).
PLC parameters	**T105**	**Tool magazine tool retract/return delay detection T105==5000** (default 5000ms).
PLC parameters	**T106**	**Tool magazine feed/reverse tool completion delay** T106=50 (default 50ms).
PLC parameters	**T107**	**Tool magazine tool retract/return completion delay** T107==50 (default 50ms).
PLC parameters	**T108**	**Tool magazine tool reversal tool return detection abnormal delay** T108==0 (default 0ms).
PLC parameters	**T109**	**Tool changer zero-to-bit signal delay detection** T109==0 (default 0ms).
PLC parameters	**T110**	**Tool magazine rotation delay detection** T110==3 000 (default 3000ms).
PLC parameters	**T111**	**Tool magazine counter in place delay detection** T111==2 000 (default 2000ms).
PLC parameters	**T112**	**Picker snap tool in place delay detection** T112==3000 (default 3000ms).
PLC parameters	**T113**	**Robot pull-down rotation rise-to-bit delay detection** T113==3 000 (default 3000ms).
PLC parameters	**T114**	**The robot arm returns to the origin point in place Delay detection** T114==3 000 (default 3000ms).
PLC parameters	**T115**	**Robot output detection delay** T115==3 000 (default 3000ms).
PLC parameters	**T116**	**Tool changer counter in place delay** T116==50 (default 50ms).
PLC parameters	**T117**	**M22 Finder detection delay** T117==50000 (default 50000ms).
PLC parameters	**C100**	**Total number of tool positions** C100 must = total number of tool magazine tool positions
PLC parameters	**C110**	**Arm motion count** C110==3
PLC parameters	**D110**	**The knife arm action count register** D110==C110 is not required
PLC parameters	**D100**	**The total number of tool positions D100=**=C100 is not entered
PLC parameters	**D101**	**Command knife number** The programming input is not modifiable
PLC parameters	**D102**	**The previous knife number of the command knife number** is not automatically calculated
PLC parameters	**D103**	**Cutterhead counting position** The automatic calculation is not modifiable
PLC parameters	**D104**	**Manual tool changer tool number** The automatic calculation cannot be modified

PLC parameters	**D105**	**Spindle current tool number** The current spindle tool number can be manually modified when it is unreasonable
PLC parameters	**D106**	**Data table rearrangement Automatic** calculations cannot be modified
PLC parameters	**D120**	**Tool magazine return / tool search address** Automatic calculation cannot be modified
PLC parameters	**D121**	**Tool changer return / tool search output results** Automatic calculations cannot be modified
PLC parameters	**D124**	**Tool magazine data table search range** The automatic calculation cannot be modified D124=D100+1
PLC parameters	**D126**	**Spindle zero speed output range (r/min)** D126==30
PLC parameters	**D000**	**Spindle tool number (tool magazine tool number retrieval header address)** After the zero point is set, D000=0, and the data in the tool number data table is automatically exchanged after normal tool change
PLC parameters	**D001**	**Cutterhead No. 1 socket set** D001==1
PLC parameters	**D002**	**Cutterhead No. 2 socket setting** D002==2
PLC parameters	**D003**	**Cutterhead No. 3 socket set** D003==3
PLC parameters	**D004**	**Cutterhead No. 4 socket setting** D004==4
...	**...**	**...**
...	**...**	**...**
...	**...**	**...**
PLC parameters	**D097**	**Cutterhead 97 pocket setting** D097==97
PLC parameters	**D098**	**Cutterhead 98 pocket setting** D099==98
PLC parameters	**D099**	**Cutterhead 99 pocket setting** D099==99

Note: C100 and D100 must be less than 100, and the total capacity of the tool magazine is 99; When C100=16, the D000-D016 data is valid; When C100=24, the D000-D024 data is valid; There should be no duplicate values in D0-D99;

Relevant M code

M16	The knife is released
M17	Tool clamping
M18	Untarget
M19	Spindle orientation

M21	Tool magazine forward
M22	Tool magazine finder
M23	ATC knife arm gripper
M24	The tool magazine is set back
M25	ATC knife arm pulls down
M26	The ATC knife arm returns to the origin
M50	The knife change begins
M51	End of knife change

Tool changer installation and related parameter settings

This ladder diagram requires a disc tool changer machine:

1. The spindle has a loose/clamping device to the tool, and the loosening/clamping device has a signal to loosen/clamp in place
2. The spindle has a positioning function
3. Tool magazine capacity is less than 100
4. The tool magazine has forward and reverse changes
5. The tool magazine has counting switch, reverse tool in place detection switch, return tool in place detection switch
6. The tool changer can be selected with or without a zero switch

If the tool changer does not have a zero switch, and each detection switch of the tool magazine is normally open, the K parameter is set

K030.1==0; K030.3==0; K030.4==0; K030.5==0;

Disc tool changer control conditions:

1. Automatic tool changer procedure steps and conditions:

1) Perform the tool change procedure M06 Txxx; To determine whether to change the tool, the machine tool lock, auxiliary lock, and spindle knife number are equal to the T command knife number, and the machine tool does not change the tool;
2) Tool change starts, spindle, cooling, modal values are canceled;
3) Automatic tool change starts; Determine whether the automatic tool change conditions are met, if they are not met, the alarm, automatic tool change conditions: the correct T command has been executed, the Z axis has been returned to zero to complete, and the tool is in the clamping state;
4) The spindle positioning is complete and the Z-axis returns to the second reference point
5) Tool magazine to find the tool, the tool magazine rotates to find the tool number, find the tool number and sense that the counter tool magazine rotation stops;
6) The tool magazine reverses the knife, and after the inverted knife is completed, the knife arm performs the first step action to grab the knife
7) After the knife grasp is completed, loosen the spindle, confirm that the knife is in place, and then execute the second step of the knife arm to pull down and rotate up
8) After the second step of the knife arm is completed, the spindle clamps the knife, and after confirming

that the knife is in place, the knife arm performs the third step action to return to the origin;

9) After confirming the completion of the return to the origin, the tool magazine returns to the tool;

10) After the tool magazine is back in place, the tool change is completed;

2. Manual control of tool magazine conditions:

1) The tool changer manual control rotation or reverse tool/tool return/ATC tool arm must be effective in the manual/handwheel mode, and the tool magazine zero operation must be effective in the mechanical zero point mode;

2) Tool magazine control inverted tool conditions: the system is in standby state, in manual/handwheel mode, tool sleeve in counter position, ATC tool arm in origin position;

3) Tool magazine control tool return conditions: the system is in standby, in the manual/handwheel mode, ATC tool arm is at the origin position;

4) Tool magazine control rotation conditions: the system is in standby, in the manual/handwheel mode, the tool sleeve is in place for the tool return;

5) Tool magazine zero return control conditions: the tool magazine is equipped with a zero switch, the system is in standby state, in the mechanical zero mode, the tool sleeve is in place for the tool return;

6) ATC knife arm control conditions: when the knife arm debugging function is turned on K30.6==1, it runs unconditionally; When the tool magazine debugging function is turned on K30.7==1, the tool sleeve is reversed in place, the spindle positioning is completed, and the Z axis is on the second reference point;

Tool magazine debugging:

Notes:

1. Confirm whether the system emergency stop is valid
2. Confirm that the tool changer line connection is normal
3. Confirm that the tool changer has no foreign objects in its range of movement
4. Make sure that the previous action is correct before proceeding to the next action
5. There is a special tool magazine control interface in the ladder diagram interface, and the interface can be entered through the F function key corresponding to [tool magazine back zero key], [tool magazine forward rotation key], [tool magazine reversal key], [forward/reverse tool key], [back/back tool key], [tool arm control key], [Z axis back to the second reference point key], [Z axis back to the third reference point key];
   To enter the control interface, press [ladder diagram] → [tool magazine control].

Tool changer manual and zero operation:

When the tool arm is in place, the tool magazine action is manually controlled to detect whether each step of the tool magazine is normal.

1. The tool changer usage function works
   A. The system default Ladder02.grp ladder diagram is a disc tool magazine ladder diagram, with a number parameter of 304==2 (requires restart);
   B. PLC parameters K1.0==1; K1.1==0;K1.3=0;

2. Manually confirm the direction of tool magazine rotation

A. Modify K30.7==1 to open the tool magazine debugging function and manually control the tool magazine condition removal

B. Press [Tool magazine back key] in manual/handwheel mode to make the tool magazine sleeve level in place, press [tool magazine forward rotation key] tool magazine to rotate in the direction of increasing the tool number, press [tool magazine reversal key] tool magazine to rotate in the direction of tool number reduction, if the rotation direction is opposite, the tool magazine count will be disordered resulting in tool exchange errors, and the phase sequence of the tool magazine motor can be solved.

Note: Tool magazine forward and reverse must be performed with the tool sleeve level, otherwise the machine will be damaged.

3. Manually confirm whether the tool changer reverse/return tool is normal
   A. Modify K30.7==1 to open the tool magazine debugging function and manually control the tool magazine condition removal
   B. Make sure the knife arm is at the origin
   C. Press the [Tool Commutor Inverted Tool Key] tool sleeve vertically in the manual/handwheel mode, press the [Tool Magazine Back Tool] tool sleeve horizontally, if the direction is opposite, you can modify the solenoid valve or change the air pipe to solve.

4. Tool magazine zero operation
   A. If the tool changer has a zero switch, modify K30.1==1;
   1) Manually return the horizontal position of the tool sleeve, and then switch to the mechanical reset mode and press [Tool magazine return to zero button] The tool magazine is turning back to zero, and at the same time issue an A20.5 tool magazine zero return warning;
   2) When the tool magazine zero return is completed, the warning in the tool magazine return to zero is canceled, and at the same time, the A15.3 tool magazine origin is set successfully, press [Reset button] to cancel the alarm, and the spindle tool number becomes 0 after the tool magazine is reset to zero.
   3) If the tool changer zero return signal is not detected after more than 5 minutes of zeroing, the system alarms A15.2 tool changer zero return detection abnormality.

   B. If the tool changer does not have a zero switch, modify K30.1==0
   1) Press [Tool magazine forward rotation key] or [tool magazine reversal key] in manual/handwheel mode to turn the tool magazine cutterhead to the No. 1 cutterhead position;
   2) Press and hold the [Machine Tool Zero Point Key] for 7 seconds, the system sends out that the A15.3 tool magazine origin is set successfully, press the [Reset button] to cancel the alarm, and the tool magazine origin setting is completed.

Warning: After the tool magazine is zeroed, the tool magazine data table will be rearranged, please reconfirm the tool number before using the tool changer to avoid collision with incorrect tool number.

5. ATC tool arm control commissioning
   A. The system switches the mechanical zeroing mode to return the Z axis to zero, (ensure that the zero point of the Z axis is above the stroke of the tool arm to avoid the tool arm colliding with the spindle during the debugging of the tool arm);

B. The system switches the manual/handwheel mode, and presses the [tool magazine back button] to make the tool sleeve horizontal;

C. Turn on the ATC knife arm debugging function K030.6==1 to remove all conditions of the knife arm control;

D. Press [Knife Arm Control Key] in the manual/handwheel mode, the knife arm performs the first step action to grab the knife, and then press the [Knife Arm Control Key], the knife arm performs the second step action to pull down and rotate 180 degrees to rise the knife, press the [Knife Arm Control Key] again, the knife arm performs the 3rd step action back to the origin, if you press [Knife Arm Control Key] again, the above 3 actions will be repeated; Check whether each step is correct, if not, check whether the ATC tool arm signal wiring is correct or the signal level selection is correct.

6. Spindle positioning angle and tool change coordinate adjustment

A. Spindle positioning angle adjustment (refer to spindle servo drive manual).

Spindle positioning angle adjustment steps (examples):

1) The system enters the operation level above level 2, open the ATC tool arm debugging function K030.6==1 The system issues an A20.4 tool magazine debugging function open warning, and the ATC tool arm can be operated unconditionally;
2) Perform Z-axis zeroing in mechanical zero mode, and wait for Z-axis zeroing to be completed (make sure that the Z-axis zero point is above the knife arm stroke to avoid the knife arm colliding with the spindle during the knife arm debugging);
3) Prepare a handle and remove the handle nails;
4) When the system switches to manual or handwheel mode, press [Knife Arm Control Key] to make the ATC knife arm perform the first step action to grab the knife;
5) Install the nailless shank on the knife arm and adjust the position;
6) The system switches to the handwheel mode, which can move the Z axis up and down (please operate with caution); At this time, directly twist the spindle, rotate for more than one week, adjust the spindle position by hand, so that the key on the spindle can enter the slot of the tool holder;
7) Record the current spindle drive display angle and set it to the drive positioning angle (refer to the drive manual);
8) The handwheel moves the Z axis up from the shank position and removes the shank on the ATC arm;
9) Press the [Blade Arm Control Key] twice to return the ATC blade arm to the original point.

B. Z-axis tool change point coordinate setting

1) After adjusting the spindle positioning according to the above operation, move the Z axis to a safe position in the manual/handwheel mode, press the [knife arm control key] to return the ATC knife arm to the original point, press the [spindle positioning key] to perform the spindle positioning, then press the [tool change key] to loosen the knife, and then press the [knife arm control key] to make the ATC knife arm perform the first step action to grab the knife, install the handle with a pull nail on the ATC knife arm to adjust the position (at this time the knife arm is in the knife grabbing position), the system switches to the handwheel mode, You can move the Z axis up and down to adjust the tool change coordinates (make sure that the tool is released at this time), and record the adjusted Z axis machine tool coordinates;
2) Move the handwheel to a safe height, remove the handle on the ATC arm, and press the [Arm Control Key] to return the ATC arm to the original position;

3) Enter the recorded Z-axis machine coordinates to the second reference point of the Z-axis [number parameter #057], this parameter must be set, otherwise the mechanical part will be damaged;
4) Turn off the ATC knife arm debugging function K30.6==0;

7. Enter to complete the tool change action debugging
   A. Open the tool magazine debugging function K30.7==1, K30.6==0; ATC tool arm control is controlled by conditions;
   B. The system switches the manual/handwheel mode and presses [Z axis return to the second reference point key] to make the Z axis return to the second reference point;
   C. The system switches the manual/handwheel mode and presses the [spindle positioning key] to complete the spindle positioning;
   D. Press [Tool magazine reverse tool key] to make the tool magazine vertical in place;
   E. Press [knife arm control key] ATC knife arm to perform the first step action to grab the knife;
   F. Press the [Tool Change Key] to loosen the spindle and loosen the knife in place
   G. Press [Blade Arm Control Button] ATC Blade Arm to perform the second step action to pull down and rotate 180 degrees up;
   H. Press [Tool Change Key] to make the spindle clamp the tool and clamp it in place
   I. Press [Blade Arm Control Key] ATC Blade Arm to perform step 3 to return to the origin;
   J. Press [Tool magazine back key] to make the tool magazine level in place;
   K. Manual tool change is done.
   L. After changing the tool correctly, turn off the tool magazine debugging K30.7==0, K30.6==0;

ATC tool arm step 1 and 3 action control conditions: spindle positioning, Z-axis at the second reference point, tool sleeve vertical, spindle tool clamping;
ATC tool arm step 2 action control conditions: spindle positioning, Z-axis at the second reference point, tool sleeve vertical, spindle tool release;

8. After passing the above operation (point 7), you can call the tool changing program O910 2 in edit mode, write the Txxx instruction at the beginning of the program, and then switch to automatic mode to run the program in a single section to check whether the tool change action is correct (modify the tool change program O9102 must put the bit parameter #029.5==0, and after the debugging is completed, put #029.5==1 to avoid mistakenly modifying the tool change program);

   The modified tool change procedure is:

   O9102;

   Txxx; **(Added during single-segment detection program, delete this segment after the tool change action is normal)**).

   G65H81P50Q#1003R1;//G54.3=1 command knife is equal to spindle knife does not perform tool change

   G69G50G15G80G40;

   M05;//Spindle off

   M09;// Cooling off

   M50;//Automatic tool change starts

   M19G00G91G49G30Z0;//Z downward spindle orientation (quick to reach the second

reference point)

M22;//Tool magazine rotary tool seeker
M21;//Tool magazine reversed knife
M23;// Arm gripper
M16;//Knife loosening
M25;//Knife arm pulls down
M17;//Spindle clamping cutter
M26;//Knife arm restoration
M24;//Tool magazine return tool
M18;// Unorient the spindle
N50M51;//Automatic tool change ends
M99;
%

9. After executing point 8, delete the Txxx command added at the beginning of the tool changer O9102, and execute the automatic tool change instruction in MDI mode after automatic mode:

① M06 Txxx or (2) Txxx ;

G54 G90;//Machining program
X100 Y100;
... ...
... ...
M06;

Perform two programs to change the tool with the same effect;
Program (2) format editing can realize pre-selection knife change;

Note: The following program is malformed:

T01;
T02;
M06;

The correct way to deal with emergency stop, reset, alarm, and power failure encountered during the operation of the tool magazine:

1. In the process of automatic tool change, the tool magazine is looking for the command tool number to complete, the spindle has dropped to the tool change position, the tool magazine has not reversed, such as emergency stop, reset, alarm, power failure, if you need to continue to use the machine tool, the reset alarm can be used normally;

2. In the process of automatic tool change, the tool magazine search for the command tool number is

completed, the spindle has dropped to the tool change position, the tool magazine is reversed to complete, the ATC tool arm has not grasped the knife, such as emergency stop, reset, alarm, power failure, if you need to continue to use the machine tool, if you need to continue to use the machine tool, you must press the [tool magazine back tool key] in the manual/handwheel mode to make the tool sleeve level, you can use it normally;

3. In the process of automatic tool change, when the tool magazine cutterhead rotates and selects the tool, in case of emergency stop, reset, alarm, power failure, there may be inconsistencies between the current tool number and the actual tool number, the system prompts A015.0 (tool magazine tool number counting signal detection is abnormal, tool magazine needs to return to zero) or A015.4 (abnormal tool change stop to ensure that the tool magazine is normal) alarm, if you need to continue to use the machine tool, reset the tool number of the tool magazine in the mechanical zero return mode (tool magazine back to zero), It can be used normally.

4. In the process of automatic tool change, when the tool magazine tool finder completes the tool arm has grabbed the knife, in case of emergency stop, reset, alarm, power failure, if you need to continue to use the machine tool, you need to adjust the tool arm back to the origin, if you manually release the tool arm motor brake, use the wrench to rotate the tool arm in the opposite direction, so that the tool arm returns to the origin of the tool, and then restore the brake, the system switches the manual/handwheel mode and presses [Tool magazine return key] to make the tool sleeve level, which can be used normally (Note: At this time, the tool has not been exchanged, and the spindle tool number has not changed). If the tool magazine debugging function is used to return the tool arm to the origin, the tool magazine knife number may be messed up, after the tool arm returns to the origin, the tool magazine needs to return to zero to be used normally, and the tool magazine knife number is rearranged, and the tool number is confirmed before use to avoid hitting the knife; Open the tool magazine debugging K30.7==1, restore the knife arm according to the above debugging steps, if the restoration is unsuccessful, turn on the knife arm debugging function K30.6==1, remove all restrictions on the knife arm, and after restoring the knife arm, you need to close the debugging function K30.7==0, K30.6==0;

5. In the process of automatic tool change, when the tool arm has been pulled down and rotated up, in case of emergency stop, reset, alarm, power failure, if you need to continue to use the machine tool, you need to adjust the tool arm back to the origin, the cutterhead number is not exchanged (tool magazine data is messy), whether you manually release the tool arm motor brake adjustment or use the tool magazine debugging function to adjust the tool arm back to the origin, after the tool magazine returns to its original state, you need to reset the cutterhead number (tool magazine to zero) to use normally.

6. In the process of automatic tool change, when the knife arm returns to the origin, in case of emergency stop, reset, alarm, power failure, if you need to continue to use the machine tool, you need to adjust the knife arm back to the origin, at this time, the spindle knife number shows the command knife number, the cutterhead knife number has been exchanged, manually release the knife arm motor brake to adjust the knife arm back to the origin, and the tool magazine can be used normally after returning to the original state.

Warning: After the tool magazine is zeroed, the tool magazine data table will be rearranged, please reconfirm the tool number before using the tool changer to avoid collision with incorrect tool number.

Disc tool changer macro program with comments

O9101; (Program Name)

G65H81P50Q#1004R1;//G54.4=1Do not perform tool change without tool change

G65H81P50Q#1003R1; (Machine tool lock, auxiliary lock, spindle tool number = T command tool number jump N50 tool change ends).

G69G50G15G80G40; (Cancel modal values).

M05;//Spindle off

M09;// Cooling off

M50; (Start from the custom tool change, and detect whether the tool change conditions are met at the same time, and alarm if it is not met).

M19G91G49G30Z0; (spindle positioning, Z-axis tool change point [2nd reference point]).

M22; (Tool magazine tool search, tool search conditions: return knife in place)

M21; (Tool magazine inverted tool, inverted tool condition: tool magazine positioning sensed, tool magazine count arrived)

M23; (knife arm gripping knife, gripping condition spindle positioning completed, Z-axis resetting tool point, knife sleeve inverted knife in place)

M16;//Knife loosening

M25; (The knife arm is pulled down, the spindle positioning of the knife arm pull-down condition is completed, the Z axis changes the tool point, the knife sleeve is reversed in place, and the knife is loosened in place).

M17;//Spindle clamping cutter

M26; (Tool arm restoration, tool arm reduction condition spindle positioning completed, Z-axis tool change point, knife sleeve inverted in place, clamping knife in place)

M24; (Tool magazine return tool, return tool condition: tool arm at the origin position)

M18;// Unorient the spindle

M51;//Automatic tool change ends

N50M99;

%

4. Turret type tool magazine

Turret tool changer application considerations

1. During the tool change process, it must be ensured that each step is completed before the next action can be carried out;
2. Before the tool magazine positioning sensor is confirmed, the Z axis cannot be lowered;
3. The spindle has not changed the tool point, and the cutterhead cannot be rotated;
4. The spindle positioning is not completed, and the tool cannot be changed;
5. The spindle cannot be rotated before the spindle comes down to clamp the knife;
6. When the voltage of the tool magazine motor is too high, the overload protection relay should trip off first to avoid burning out the tool magazine motor;

7. Please change the tool manually first to check that the forward and reverse movements of the tool magazine are correct, and then start the CNC automatic sequence program control;
8. During the debugging process, only the input and output signals (X/Y address) of the ladder diagram need to be modified to complete the debugging requirements; If you are not sure, do not modify the others to avoid accidents.

Relevant signals

Foot size	address	definition	remark
CN61.11	X1.2	Tool changer return to zero to place signal	Level modification K30.2
CN61.31	X2.2	Tool changer counting switching signals	Level modification K30.5
CN15.08	X4.2	The spindle is directed to the signal in place	Level modification K21.1
CN61.38	Y3.1	Tool changer forward output	M22
CN61.39	Y3.2	The tool changer reverses the output	M22
CN15.22	Y4.2	Spindle directional output	M19/M18

Note: M22 cannot be used alone.

parameter	**serial number**	**illustrate**
Number parameters	**#052**	**Z-axis 1st reference point value** defaults to 0
Number parameters	**#057**	**Z-axis 2nd reference point value (downward tool change point).**
Number parameters	**#062**	**Z-axis 3rd reference point value (upward tool change point).**
Number parameters	**#226**	**Tool magazine allowable rotation range minimum (tool magazine allowable rotation interval) F23.4** Z-axis machine coordinates
Number parameters	**#227**	**The maximum allowed rotation range of** the tool changer
Number parameters	**#228**	**Minimum value of** the slow **tool change interval** (**slow** change **interval) F23.5** Z-axis machine coordinates
Number parameters	**#229**	**The maximum value** of the slow **range** for tool change
bit parameters	**#23.5**	**Change the tool slow zone speed limit (0: invalid 1: valid)** to 1
Number parameters	**#216**	**Z-axis tool change slow range limits speed F** to 1000 default

Note: If the turret changer needs to pass through the slow interval at a slow speed during the tool change interval, open the turret tool changer slow speed limit function #23.5=1, and the speed is controlled by the numerical value set by parameter #216, and this parameter controls G 0/G53/G28/G30 speed through the tool change interval; When the speed of G 0/G53/G28/G30 is lower than the speed set by parameter #216, pass through the tool change slow interval at the speed of G 0/G53/G28/G30 ； If the tool changer does not need to pass slowly in the tool change interval, do not turn on the change speed limit function #23.5=0.		
Number parameters	**#236**	**Minimum F23.6** Z-axis machine coordinates for the forbidden range of the turret changer spindle
Number parameters	**#237**	**The maximum value** of the Turret changer spindle forbidden interval is 9999
Number parameters	**#238**	**Minimum F23.7** Z-axis machine coordinates for the tool change interval of the turret change
Number parameters	**#239**	**The maximum value** of the turret changer is 9999
Note: Only after the spindle positioning is completed can it enter the spindle prohibition range; Note: When entering the tool change range, the manual/handwheel mode button triggers the spindle positioning is invalid; Note: When the manual/handwheel enters the turret tool change interval, the spindle positioning signal is triggered (provided that K32.7=1) is triggered NOTE: When running commands other than the tool change program to enter the turret tool change range, alarm A16.4 is triggered		
PLC parameters	**K001.0**	**(0: Forbidden 1: Allow) Tool changer use** Select 1 when using the tool changer
PLC parameters	**K001.1**	**Tool magazine usage (0: other tool changers 1: row of tools)** 0 is selected when the tool row is not in use
PLC parameters	**K001.3**	**When the tool changer function is turned on (0: use 1: do not use) the tool changer** is selected when using the tool changer
PLC parameters	**K014.6**	**(0: Check 1: Do not check) Tool clamping/loosening in place signal** Turret tool changer selection 1
PLC parameters	**K021.1**	**The spindle positioning completion detection signal is (0:normally open 1:normally closed) is valid**
PLC parameters	**K030.1**	**Tool changer (0:No 1:Yes) with a return switch** Select 1 when there is a zero switch
PLC parameters	**K030.2**	**The tool changer zero switch signal is (0:NO 1:NC) active**
PLC parameters	**K030.5**	**The tool magazine count switch signal is (0:normally open 1:normally closed) active**
PLC parameters	**K030. 6**	**Disc magazine arm/turret (0:No 1:Yes) enters debug mode**
Note: When opening the turret to debug K30.6=1, the Z axis is not restricted by conditions, and it is operated carefully; Note: When shielding soft limit K 32.4=1 during tool change, when the debugging function K30.7=1 is turned on, the soft limit can be shielded;		

PLC parameters	**K030.7**	**The tool changer (0:No 1:Yes) enters debug mode**
PLC parameters	**K032.4**	**During tool change (0:No 1:Yes) shield soft stops**
The turret changer does not require this parameter to be set to 0.		
PLC parameters	**K032.5**	**A warning is prompted when** the spindle has **a front** tool number of 0 **(0:Yes 1:No).**
This parameter only prompts the spindle that the current tool number is 0, and warns not to install the knife on the spindle.		
PLC parameters	**K032.6**	**The changer cutterhead rotation (0:No 1:Yes) has only forward rotation**
PLC parameters	**K032.7**	**The turret changer spindle enters the tool change section (0:no 1:Yes) to automatically output positioning**
When this parameter is set to 1, the manual/handwheel Z-axis enters the tool change range, and the spindle automatically outputs positioning.		
PLC parameters	**T004**	**Spindle orientation completion delay** T004==3 0 (default 30ms).
PLC parameters	**T018**	**Spindle orientation in place signal delay detection** T018==2000 (default 2000ms).
PLC parameters	**T023**	**Spindle orientation in place detection delay** T023==10000 (default 10000ms).
PLC parameters	**T100**	**Tool changer manual forward key delay** T100==0 (default 0ms).
PLC parameters	**T101**	**Tool changer manual reverse key delay** T101==0 (default 0ms).
PLC parameters	**T102**	**Tool changer rotation stop delay detection** T102==0 (default 0ms).
PLC parameters	**T103**	**Tool magazine counter delay detection** T103==2000 (default 2000ms).
PLC parameters	**T108**	**Tool changer counting switch delay detection** T108==0 (default 0ms).
PLC parameters	**T109**	**Tool changer zero-to-bit signal delay detection** T109==0 (default 0ms).
PLC parameters	**T110**	**Tool magazine rotation delay detection** T110==3 000 (default 3000ms).
PLC parameters	**T111**	**Tool magazine counter in place delay detection** T111==2 000 (default 2000ms).
PLC parameters	**T116**	**Tool changer counter in place delay** T116==50 (default 50ms).
PLC parameters	**T117**	**M22 Finder detection delay** T117==800 0 (default 8000ms).

PLC parameters	**C100**	**Total number of tool positions** C100 must = total number of tool magazine tool positions
PLC parameters	**D100**	**The total number of tool positions** D100=C100 is not entered
PLC parameters	**D101**	**Command knife number** The programming input is not modifiable
PLC parameters	**D102**	**The previous knife number of the command knife number** is not automatically calculated
PLC parameters	**D103**	**Cutterhead counting position** The automatic calculation is not modifiable
PLC parameters	**D104**	**Manual tool changer tool number** The automatic calculation cannot be modified
PLC parameters	**D105**	**Spindle current tool number** The current spindle tool number can be manually modified when it is unreasonable
PLC parameters	**D120**	**Tool magazine return / tool search address** Automatic calculation cannot be modified
PLC parameters	**D121**	**Tool changer return / tool search output results** Automatic calculations cannot be modified
PLC parameters	**D124**	**Tool magazine data table search range** The automatic calculation cannot be modified D124=D100+1
PLC parameters	**D126**	**Spindle zero speed output range (r/min)** D126==30
PLC parameters	**D000**	**Cutterhead No. 0 socket setting (tool magazine tool number retrieval header address)** D000==100
PLC parameters	**D001**	**Cutterhead No. 1 socket set** D001==1
PLC parameters	**D002**	**Cutterhead No. 2 socket setting** D002==2
PLC parameters	**D003**	**Cutterhead No. 3 socket set** D003==3
PLC parameters	**D004**	**Cutterhead No. 4 socket setting** D004==4
...	**...**	**...**
...	**...**	**...**
...	**...**	**...**
PLC parameters	**D097**	**Cutterhead 97 pocket setting** D097==97
PLC parameters	**D098**	**Cutterhead 98 pocket setting** D099==98
PLC parameters	**D099**	**Cutterhead 99 pocket setting** D099==99

Note: C100 and D100 must be less than 100, and the total capacity of the tool magazine is 99; When C100=16, the D000-D016 data is valid; When C100=24, the D000-D024 data is valid.

Relevant M code

M18	Untarget
M19	Spindle orientation
M22	Tool magazine finder
M50	The knife change begins
M51	End of knife change

Tool changer installation and related parameter settings

This ladder diagram requires a turret tool magazine machine:

1. The spindle can choose whether there is a positioning function, if there is no positioning function, the M19/M18 in the tool changer O9101 needs to be deleted
2. Tool magazine capacity is less than 100
3. The tool magazine has forward and reverse changes
4. The tool changer has a counting switch
5. The tool changer can be selected with or without a zero switch

If the tool changer does not have a zero switch, and each detection switch of the tool magazine is normally open, the K parameter is set

K030.1==0; K030.5==0;

Turret tool changer control conditions:

1. **Automatic tool changer procedure steps and conditions:**
   1) Perform the tool change procedure M06 Txxx; To determine whether to change the tool, the machine tool lock, auxiliary lock, and spindle knife number are equal to the T command knife number, and the machine tool does not change the tool;
   2) Tool change starts, spindle, cooling, modal values are canceled;
   3) Automatic tool change starts; Determine whether the automatic tool change conditions are met, if they are not met, the alarm, automatic tool change conditions: the correct T command has been executed, the Z axis has returned to zero to complete, and the tool is in the clamping state;
   4) The spindle rises to the tool change point and returns the knife;;
   5) Tool magazine to find the tool, the tool magazine rotates to find the command T tool number, find the knife number and sense the counter;
   6) spindle lowering clamp cutter;
   7) The knife change is complete;

2. **Manual control of tool magazine conditions:**
   1) The manual control rotation of the tool magazine must be effective in the manual/handwheel mode, and the tool magazine zero operation must be effective in the mechanical zero mode;
   2) The tool magazine controls the rotation conditions, the system is in standby state, in the manual/handwheel mode, the Z axis is within the allowable rotation range of the tool magazine;
   3) Tool magazine zero control conditions, tool magazine equipped with zero switch, the system in standby

state, in the mechanical zero mode, Z axis in the tool magazine allowed rotation range.

Tool magazine debugging:

Notes:

1. Confirm whether the system emergency stop is valid
2. Confirm that the tool changer line connection is normal
3. Confirm that the tool changer has no foreign objects in its range of movement
4. There is a special tool magazine control interface in the ladder diagram interface, and the interface can be entered through the F function key corresponding to [tool magazine back zero key], [tool magazine forward rotation key], [tool magazine reversal key], [forward/reverse tool key], [back/back tool key], [tool arm control key], [Z axis back to the second reference point key], [Z axis back to the third reference point key];
   To enter the control interface, press [ladder diagram] → [tool magazine control].

Tool changer manual and zero operation:

Manually control the tool magazine action to detect whether each step of the tool magazine is normal.

1. The tool changer usage function works
   A. The system default Ladder03.grp ladder diagram is the turret tool magazine ladder diagram, with the number parameter 304==3 (requires restart);
   B. PLC parameters K1.0==1;K1 1==0； K1.3==0;

2. Manually confirm the direction of tool magazine rotation
   A. Modify K30.7==1 to open the tool magazine debugging function and manually control the tool magazine condition removal
   B. Modify K30.6==1 Turn on the turret tool magazine debugging function, the Z axis up and down movement condition is released, and the operation is careful to avoid damage to the tool magazine;
   C. Z axis back to the zero point, after the completion of the zero return, operate the spindle positioning (note that the zero point of the Z axis should be under the tool magazine), after the positioning is completed, the handwheel moves the Z axis up and down, and records the maximum and minimum value of the spindle forbidden rotation (spindle forbidden range);
   D. record the maximum and minimum allowed rotation of the tool magazine (the allowable rotation interval of the tool magazine);
   E. At the same time, a point in the tool magazine rotation interval is recorded as the tool change point at the third reference point of the Z axis of parameter #62;
   F. Record the completion point of tool change, recorded at the second reference point of the Z axis of the number parameter #57, which can coincide with the zero point of the Z axis machine tool;
   G. If you need to pass slowly when passing through the clamping knife area when changing the Z axis, you need to set the maximum and minimum values of the slow speed interval of the tool change, and at the same time need to turn on the function of limiting the speed of the slow interval (parameter #23.5; number parameter 216); If you need the spindle automatic positioning function when the manual/handwheel is moved to the tool change interval, you need to set the maximum and minimum value of the tool change interval and open the turret tool magazine spindle automatic positioning function K 32.7=1, when the Z axis enters the section, the spindle positioning signal will be sent out

(provided that K32.7=1), and the minimum value of the tool change interval is 1 0mm smaller than the minimum value of the spindle prohibition interval The above reserved spindle positioning buffer distance (the Z axis will not move and stop during the spindle positioning process), so to avoid entering the spindle prohibition interval before the spindle positioning is completed, a certain buffer distance is required.

H. After the Z axis is moved to the tool magazine to allow the rotation range, press the [tool magazine forward rotation key] tool magazine to rotate in the direction of increasing the tool number in the manual/handwheel mode, press the [tool magazine reversal key] to rotate in the direction of decreasing the tool number, if the rotation direction is opposite, the tool magazine count will be disordered resulting in tool exchange errors, and the phase sequence of the tool magazine motor can be solved.

I. After setting the coordinate values of the interval and tool change point, you must change K30.6 to 0 to turn on the Z-axis restriction on the tool change interval.

3. Tool magazine zero operation

A. When the tool magazine has a zero switch, modify K30.1==1; modify K30.7==1 to open the tool magazine debugging function and manually control the tool magazine condition to be released

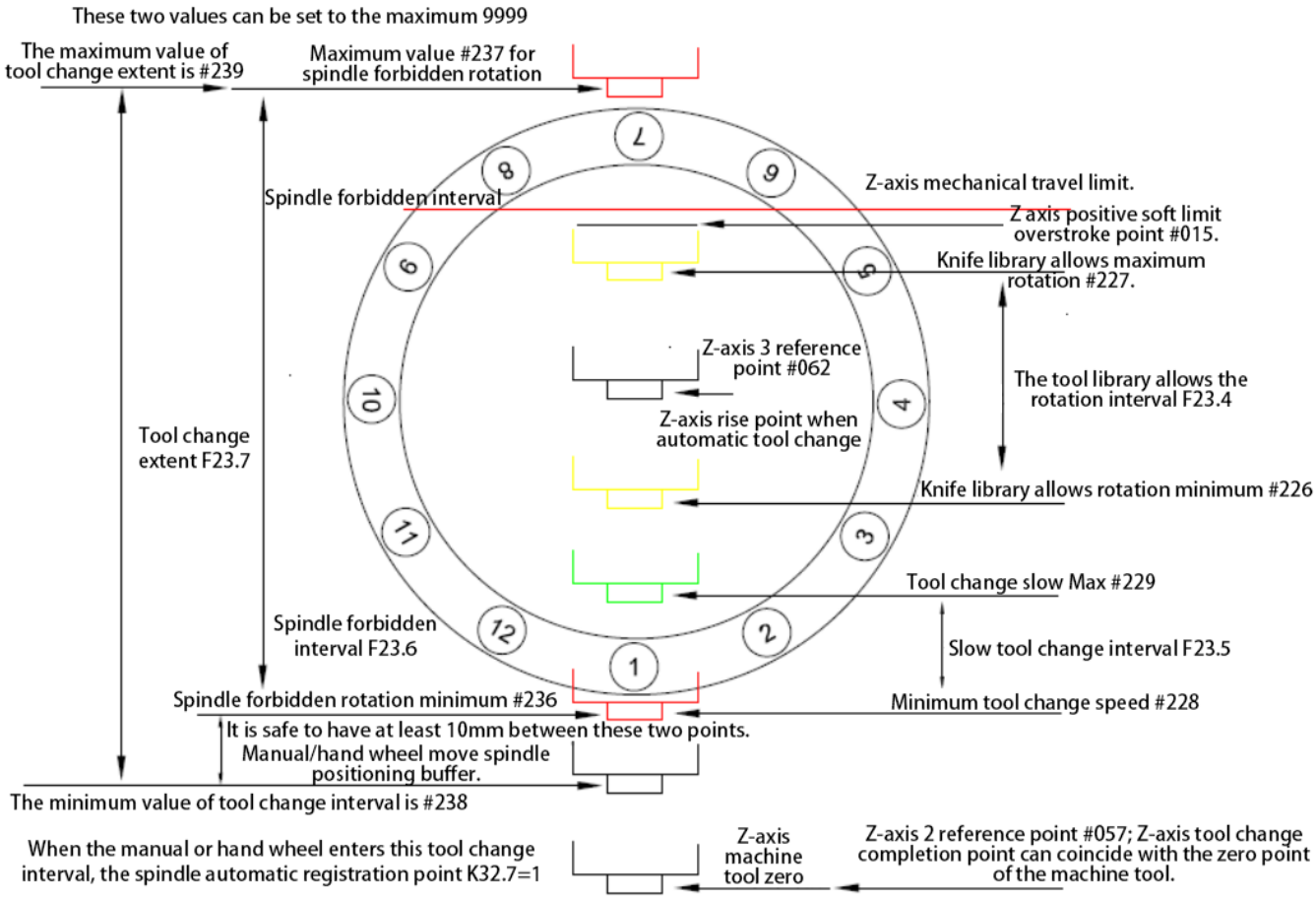
1) The Z-axis returns to the tool magazine to allow rotation to be rotated
2) Switch to the mechanical reset mode and press [Tool Change Zero Button] The tool magazine is turning back to zero, and at the same time issue an A20.5 tool magazine zero return warning;
3) When the tool magazine zero return is completed, the tool magazine zero return warning is canceled, and at the same time, the A15.3 tool magazine origin setting is successful, press the [reset button] to cancel the alarm, the tool magazine zero return is completed, and the spindle tool number changes to No. 1 tool after the tool magazine is reset to zero.
4) If the tool changer zero return signal is not detected after more than 5 minutes of zeroing, the system alarms A15.2 tool changer zero return detection abnormality.

B. If the tool changer does not have a zero switch, modify K30.1==0

1) Press [Tool magazine forward rotation key] or [tool magazine reversal key] in manual/handwheel mode to turn the tool magazine cutterhead to the No. 1 cutterhead position;
2) Press and hold the [Machine Tool Zero Point Key] for 7 seconds, the system sends out that the A15.3 tool magazine origin is set successfully, press the [Reset button] to cancel the alarm, and the tool magazine origin setting is completed.

Note: After the tool magazine is zeroed, the cutterhead number and spindle tool number are both No. 1

4. Tool change coordinate adjustment



The relationship between the maximum and minimum values of each interval is shown in the figure above

A. Z-axis upward tool feed point coordinate setting:

1) Enter the recorded Z-axis machine coordinates into parameter #062, this parameter must be set, otherwise the mechanical part will be damaged (make sure that the spindle is within the allowable rotation range of the tool magazine);

B. Z-axis downward tool change coordinate setting:

1) The handwheel moves the Z axis to descend to ensure that the tool has been clamped, record the Z axis coordinates, and enter the recorded coordinate value into the number parameter #0No. 57 (which can coincide with the zero point of the machine tool);

C. At the same time, set the maximum and minimum value of the forbidden rotation interval of the spindle and the maximum and minimum value of the allowable rotation interval of the tool magazine; The spindle cannot be rotated in the forbidden range of the spindle, and the tool magazine can only rotate within the allowable rotation interval of the tool magazine.

5. After passing the above two steps (point 4), you can call the tool change program O910 3 in edit mode, write the Txxx command at the beginning of the program, and then switch to automatic mode to run the program in a single section to check whether the tool change action is correct (modify the tool change program O9103 must put the bit parameter #029.5==0, and #029.5==1 after debugging is completed to avoid mistakenly modifying the tool change program);

The modified tool change procedure is:

O9103;

Txxx; **(Added during single-segment detection program, delete this segment after the tool change action is normal)**).

G65H81P50Q#1003R1;//G54.3=1 command knife is equal to spindle knife does not perform tool change

G65H81P50Q#1002R1;//G54.2 command knife is 0

G69G50G15G80G40;

M05;//Spindle off

M09;// Cooling off

M50;//Automatic tool change starts

G65H81P20Q#1000R1;//G54.0=1 Spindle tool number is 0

M19G91G49G30Z0; The spindle is positioned upward (quickly to the second reference point).

G91G49G30Z0P3;//Z axis upward spindle orientation (fast to the third reference point).

M22;//Tool magazine rotation

G91G49G30Z0;//Z axis downlink (fast to second reference point)

M18;// Unorient the spindle

N50M51;//Automatic tool change ends

M99;

%

6. After executing point 5, delete the Txxx command added at the beginning of the tool changer O9103, and execute the automatic tool change instruction in MDI mode after automatic mode:

Txxx M06 or M06 Txxx ;

Warning: When the spindle tool number is 0, the spindle is forbidden to load the knife, otherwise it will malfunction during automatic tool change, resulting in mechanical damage!

The correct way to deal with emergency stop, reset, alarm, and power failure encountered during the operation of the tool magazine:

In the process of automatic tool change, the tool magazine to find the command knife number is completed, the spindle is not down the clamping tool, in case of emergency stop, reset, alarm, power failure, if you need to continue to use the machine tool, move the Z-axis descending clamping knife in the manual/handwheel mode (at this time, the spindle knife number is changed to the command knife number), it can be used normally;

In the process of automatic tool change, the Z axis rises to the tool change point to loosen the tool, the cutterhead has not yet rotated, in case of emergency stop, reset, alarm, power failure, if you need to continue to use the machine tool, you must move the Z axis descending clamping tool in the manual / handwheel mode (at this time the spindle tool has not changed) can be used normally;

In the process of automatic tool change, the Z axis rises to the tool change point to loosen the tool, the cutterhead rotates the tool selection but the tool selection is not completed, in case of emergency stop, reset, alarm, power failure, there may be inconsistencies between the current tool number and the actual tool number, the system will prompt A015.0 (tool magazine tool number counting signal detection is abnormal, tool magazine needs to return to zero) or A015.4 (abnormal tool change stop to ensure that the tool magazine is normal) alarm, (at this time the spindle tool number is 0) If you need to continue to use the machine tool, reset the tool magazine tool number (tool magazine back to zero, After the tool magazine is reset to zero, the spindle tool number and the current tool magazine of the tool magazine are both No. 1), and then move the Z-axis descending clamp tool in manual/handwheel mode to use it normally.

Turret tool changer macro program with comments

```
O9103;                    (program name).
G65H81P50Q#1004R1;//G54.4=1Do not perform tool change without tool change
G65H81P50Q#1003R1; (Machine tool lock, auxiliary lock, spindle tool number = T command tool
number when jumping N50 tool change end)
G65H81P50Q#1002R1;//G54.2 command knife is 0
G69G50G15G80G40;      (Cancellation of relevant modal values).
M05;//Spindle off
M09;// Cooling off
M50;                      (Start from customizing the tool change, and detect whether the tool change
conditions are met, and alarm if it is not met)
M19G91G49G30Z0;   (Spindle positioning uplink [quick to reach 2nd reference point]).
G91G49G30Z0P3;      (Spindle positioning, Z-axis tool change point [3rd reference point]).
M22;                      (Finder-> tool magazine rotary tool finder)
G91G49G30Z0P2;      (Spindle downward clamping tool [quick to reach the 2nd reference point])).
M18;// Unorient the spindle
N50M51;//Automatic tool change ends
M99;
%
```

5. Servo bucket type tool magazine

Relevant signals

Foot size	address	definition	remark
CN61.03	X0.2	Manual loose/clamp control	Normally open contacts
CN61.09	X1.0	The knife is released into place signal	Normally open contacts
CN61.10	X1.1	Tool clamping in place signal	Normally open contacts
CN61.29	X2.0	Tool changer forward in place signal	Level modification K30.3
CN61.30	X2.1	Tool changer back into place signal	Level modification K30.4
CN61.35	X2.6	The current cutterhead has a tool detection signal	Level modification K32.3
CN61.36	X2.7	The spindle has a knife detection signal	Level modification K32.1
CN15.08	X4.2	The spindle is directed to the signal in place	Level modification K21.1
CN61.03	Y0.2	Tool loosening/clamping output	M16/M17
CN61.40	Y3.3	Tool changer forward output	M21
CN61.41	Y3.4	The tool changer outputs backwards	M24
CN61.42	Y3.5	Tool changer back to position indicator	
CN15.22	Y4.2	Spindle directional output	M19/M18

Note: M21/M22/M24 cannot be used alone.

Description of the relevant parameters

parameter	**serial number**	**illustrate**
Number parameters	**#052**	**Z-axis 1st reference point value (default to 0).**
Number parameters	**#057**	**The 2nd** reference point value of the Z **axis (tool change downward point loading point).**
Number parameters	**#062**	**Z-axis 3rd reference point value (loosening knife up point).**
Note: The reference point of Z axis 1 is set to 0 by default, the second reference point is the tool magazine tool loading position point, and the third reference point is the main axis after the main axis is loosened, and the spindle rises the tool seeking position point. (All are machine coordinate values)		

bit parameters	**#23.5**	**Z-axis tool change slow interval degree limit (0: invalid 1: effective).**
bit parameters	**#23.6**	**X-axis tool change slow interval speed limit (0: invalid 1: valid).**
bit parameters	**#23.7**	**Y-axis tool change slow interval speed limit (0: invalid 1: valid).**
Number parameters	**#216**	**Z-axis tool change slow range limits speed F** to 1000 default
Number parameters	**#218**	**X-axis tool change slow interval limits speed F** default to 1000
Number parameters	**#219**	**Y-axis tool change slow range limits speed F** default to 1000
Number parameters	**#228**	**Z-axis tool change slow speed interval minimum (tool change slow interval) F23.5** Z-axis machine coordinates
Number parameters	**#229**	**The maximum value of the Z-axis tool change slow interval**
Number parameters	**#367**	**Minimum value of X-axis tool change slow interval (tool change slow interval) F23 2** X-axis machine coordinates
Number parameters	**#368**	**The maximum value of the X-axis tool change slow interval**
Number parameters	**#380**	**Y-axis tool change slow interval minimum (tool change slow interval) F23 2** Y-axis machine coordinates
Number parameters	**#381**	**The maximum value of the Y-axis tool change slow interval**
Note: If the tool changer needs to pass the slow speed at a slow speed during the tool change interval, turn on the tool change slow speed limit function #23.5=1 or #23.6=1 or #23.7=1 (depending on which axis is required for the tool magazine), and the speed is determined by the number parameter #216 or #218 or #219 set numerical control, this parameter controls the speed of G 0/G53/G28/G30 through the tool change interval; When the speed of G 0/G53/G28/G30 is lower than the speed set by parameter #216, pass through the tool change slow interval at the speed of G 0/G53/G28/G30 ； If the tool changer does not need to pass slowly in the tool change interval, do not turn on the change speed limit function #23.5=0 or #23 6=0 or #23 7=0。		
Memorize variables	**#900**	**Coordinates of the No. 1 tool machine (X-axis).**
Memorize variables	**#901**	**Coordinates of the No. 1 tool machine (Y/A axis).**
Memorize variables	**#902**	**Coordinates of the No. 2 tool machine (X-axis).**
Memorize variables	**#903**	**Coordinate of the No. 2 tool machine (Y/A axis).**
Memorize variables	**#904**	**Coordinates of the No. 3 tool machine (X-axis).**
Memorize variables	**#905**	**Coordinate of the No. 3 tool machine (Y/A axis).**

Memorize variables	**#906**	**Coordinates of the No. 4 tool machine (X-axis).**
Memorize variables	**#907**	**Coordinates of the No. 4 tool machine (Y/A axis).**
Memorize variables	**#908**	**Coordinates of the tool No. 5 tool (X-axis).**
Memorize variables	**#909**	**Coordinates of the No. 5 tool machine (Y/A axis).**
Memorize variables	**...**	**...**
Memorize variables	**...**	**...**
Memorize variables	**...**	**...**
PLC parameters	**K001.0**	**(0: Forbidden 1: Allowed) Tool magazine use** Select 1 when using row tools
PLC parameters	**K001.1**	**Tool magazine usage (0: other tool changers 1: row of tools) Select** 1 when using tool row tools
PLC parameters	**K001.3**	**When the tool changer function is turned on (0: use 1: do not use) the tool changer** is selected when using the tool changer
PLC parameters	**K014.6**	**(0: Check 1: Do not check) Tool clamping/loosening in place signal** Select 1 when no signal is received
PLC parameters	**K014.0**	**The spindle release signal is (0: normally open 1: normally closed) is valid** and 0 is selected for normally open
PLC parameters	**K014.1**	**The spindle clamping signal is (0: normally open 1: normally closed) is valid** and 0 is selected for normally open
PLC parameters	**K021.1**	**The spindle positioning completion detection signal is (0:normally open 1:normally closed) is valid**
PLC parameters	**K030.3**	**The tool magazine forward/reverse tool in place signal is (0:normally open 1:normally closed) is valid**
PLC parameters	**K030.4**	**The tool magazine back/return signal is (0:normally open 1:normally closed) is valid**
PLC parameters	**K030.7**	**The tool changer (0:No 1:Yes) enters debug mode**
PLC parameters	**K031. 0**	**When there is no tool after tool clamping (0:Yes 1:No), the clamping signal can be detected**
This parameter applies to spindles that do not detect the tool clamping signal when the spindle is not loaded, and this parameter can be set to 1 when this spindle is such a spindle.		
PLC parameters	**K031.2**	**Output after the tool changer is forward/reversed in place (0: uninterrupted 1: disconnected).**
PLC parameters	**K031.3**	**Output after the tool magazine is back/back in place (0: uninterrupted 1: disconnected).**
PLC	**K032.0**	**Spindle with knife detection (0: invalid 1: valid)**

parameters		
See whether the machine tool has a tool detection signal for the spindle installed, generally not installed, select 0		
PLC parameters	**K032.1**	**The spindle tool detects that the switch signal is (0:normally open 1:normally closed) active**
PLC parameters	**K032.2**	**Current cutterhead changer with cutter detection (0: invalid 1: valid)**
See whether the machine tool has an installed cutterhead with a tool detection signal, generally not installed, select 0		
PLC parameters	**K032.3**	**The current cutterhead detection switch signal is (0:NO 1:NC) active**
PLC parameters	**K032.4**	**During tool change (0:No 1:Yes) shield soft stops**
The tool changer does not need to set this parameter and is 0.		
PLC parameters	**K032.5**	**A warning is prompted when** the spindle has **a front** tool number of 0 **(0:Yes 1:No).**
This parameter only prompts the spindle that the current tool number is 0, and warns not to install the knife on the spindle.		
PLC parameters	**T007**	**Tool release completion delay** can be appropriately delayed (default 30ms).
PLC parameters	**T008**	**Tool clamping completion delay** can be appropriately delayed (default 30ms).
PLC parameters	**T009**	**Spindle tool release detection delay** T009==5000 (default 5000ms).
PLC parameters	**T020**	**Spindle tool clamping detection delay** T020==5000 (default 5000ms).
PLC parameters	**T004**	**Spindle orientation completion delay** T004==3 0 (default 30ms).
PLC parameters	**T018**	**Spindle orientation in place signal delay detection** T018==2000 (default 2000ms).
PLC parameters	**T023**	**Spindle orientation in place detection delay** T023==10000 (default 10000ms).
PLC parameters	**T104**	**Tool magazine tool feed/reverse tool delay detection** T104==5000 (default 5000ms).
PLC parameters	**T105**	**Tool magazine tool retract/return delay detection T105==5000** (default 5000ms).
PLC parameters	**T106**	**Tool magazine feed/reverse tool completion delay** T106==50 (default 50ms).
PLC parameters	**T107**	**Tool magazine tool retract/return completion delay** T107==50 (default 50ms).

PLC parameters	**T108**	**Tool change forward and backward detection abnormal delay** T108==0 (default 0ms).
PLC parameters	**C100**	**Total number of tool positions** C100 must = total number of tool magazine tool positions
PLC parameters	**D100**	**The total number of tool positions** D100=C100 is not entered
PLC parameters	**D101**	**Command knife number** The programming input is not modifiable
PLC parameters	**D102**	**The previous knife number of the command knife number** is not automatically calculated
PLC parameters	**D103**	**Cutterhead counting position** The automatic calculation is not modifiable
PLC parameters	**D104**	**Manual tool changer tool number** The automatic calculation cannot be modified
PLC parameters	**D105**	**Spindle current tool number** The current spindle tool number can be manually modified when it is unreasonable
PLC parameters	**D120**	**Tool magazine return / tool search address** Automatic calculation cannot be modified
PLC parameters	**D121**	**Tool changer return / tool search output results** Automatic calculations cannot be modified
PLC parameters	**D124**	**Tool magazine data table search range** The automatic calculation cannot be modified D124=D100+1
PLC parameters	**D126**	**Spindle zero speed output range (r/min)** D126==30
PLC parameters	**D000**	**Cutterhead No. 0 Pocket Setting (Tool Change Number Retrieval Table Address)** D000==0
PLC parameters	**D001**	**Cutterhead No. 1 socket set** D001==1
PLC parameters	**D002**	**Cutterhead No. 2 socket setting** D002==2
PLC parameters	**D003**	**Cutterhead No. 3 socket set** D003==3
PLC parameters	**D004**	**Cutterhead No. 4 socket setting** D004==4
...	**...**	**...**
...	**...**	**...**
...	**...**	**...**
PLC parameters	**D097**	**Cutterhead 97 pocket setting** D097==97
PLC parameters	**D098**	**Cutterhead 98 pocket setting** D099==98
PLC parameters	**D099**	**Cutterhead 99 pocket setting** D099==99

Relevant M code

M16	The knife is released
M17	Tool clamping
M18	Untarget
M19	Spindle orientation
M21	Tool magazine forward
M22	Tool magazine forward
M24	The tool magazine is set back
M50	The knife change begins
M51	End of knife change

Tool changer installation and related parameter settings

This ladder diagram requires the servo tool changer machine:

1. The spindle has a tool loosening/clamping device, and the loosening/clamping device can be selected with or without a loosening/clamping in place signal
2. The spindle can choose whether it has a positioning function, if there is no positioning function, the M19/M18 in the tool changer O9104 needs to be deleted
3. Tool magazine capacity is less than 100
4. The tool changer uses a servo motor to control the angle
5. Forward in position detection switch, back in position detection switch

When each detection switch of the tool magazine is normally open, the K parameter is set
K030.3==0; K030.4==0;

Servo tool magazine control conditions:

1. **Automatic tool changer procedure steps and conditions:**
   1) Perform the tool change procedure M06 Txxx; To determine whether to change the tool, the machine tool lock, auxiliary lock, and spindle knife number are equal to the T command knife number, and the machine tool does not change the tool;
   2) Tool change starts, spindle, cooling, modal values are canceled;
   3) Automatic tool change starts; Determine whether the automatic tool change conditions are met, if they are not met, the alarm, automatic tool change conditions: the correct T command has been executed, the Z axis has been returned to zero to complete, and the tool is in the clamping state;
   4) Determine whether to skip the return part, jump when the spindle knife number is 0 or the spindle has no knife;
   5) The tool magazine returns the tool, the tool magazine rotates to find the tool number to be returned, and the tool magazine advances after moving to the corresponding coordinates;
   6) spindle loosening knife;
   7) Determine whether to skip the tool hunting part, and jump when the command knife number Txxx is T0;
   8) Tool magazine to find the tool, the tool magazine rotates to find the command T tool number, and the

tool magazine advances after moving to the corresponding coordinates;

9) spindle clamping knife;

10) The tool magazine is retreated, and the spindle can only be retreated after the knife is clamped;

2. **Manual control of tool magazine conditions:**

1) Manual control of the tool magazine forward and backward must be effective in the manual/handwheel mode;

2) The tool magazine controls the forward conditions, the system is in standby mode, and the tool magazine is in the debugging state K30.7=1 in the manual/handwheel mode;

3) Tool magazine control back condition, the system in standby state, in manual/handwheel mode, tool clamping state;

Tool magazine debugging:

Notes:

1. Confirm whether the system emergency stop is valid
2. Confirm that the tool changer line connection is normal
3. Confirm that the tool changer has no foreign objects in its range of movement
4. There is a special tool magazine control interface in the ladder diagram interface, and the interface can be entered through the F function key corresponding to [tool magazine back zero key], [tool magazine forward rotation key], [tool magazine reversal key], [forward/reverse tool key], [back/back tool key], [tool arm control key], [Z axis back to the second reference point key], [Z axis back to the third reference point key];
   To enter the control interface, press [ladder diagram] → [tool magazine control].

Tool changer manual operation:

Manually control the tool magazine action to detect whether each step of the tool magazine is normal.

1. The tool changer usage function works
   A. The system default Ladder0 4.grp ladder diagram is the bucket tool magazine ladder diagram, the number parameter 304==4 (need to restart);
   B. PLC parameters K1.0==1;K1.1==1; K1.3==0；

2. Manually confirm that the gear ratio of the rotary shaft of the tool changer is correct
   A. Modify K30.7==1 to open the tool magazine debugging function and manually control the tool magazine condition removal
   B. In the manual/handwheel mode, the tool changer servo motor shaft is controlled by movement to check whether the gear ratio is correct.

3. Manually confirm whether the tool changer forward and backward is normal
   A. Modify K30.7==1 to open the tool magazine debugging function and manually control the tool magazine condition removal
   B. The Z-axis return to zero ensures that the tool changer is pushed out without colliding with the spindle

C. Press the [Tool Magazine Forward Key] tool magazine in the manual/handwheel mode, press the [Tool Magazine Back Button] tool magazine to retract, if the direction is opposite, you can modify the solenoid valve or change the air change pipe to solve.

4. Spindle positioning angle and tool change coordinate adjustment

A. Spindle positioning angle adjustment (refer to spindle servo drive manual).

Spindle positioning angle adjustment steps (examples):

1) The system enters the operation level above level 2, open the tool magazine debugging function K30.7==1 The system issues an A20.4 tool magazine debugging function opening warning, and the tool magazine can be unconditionally entered and retreated;
2) Perform Z-axis zeroing in mechanical zero-point mode, and wait for Z-axis zeroing to be completed;
3) Prepare a handle and remove the handle nails;
4) The system switches to manual or handwheel mode, press [Tool magazine forward button] to make the tool magazine in the forward position of the tool magazine;
5) Install the nailless shank on the current cutterhead of the tool magazine and adjust the position;
6) The system switches to the handwheel mode, which can move the Z axis up and down (please operate with caution); At this time, directly twist the spindle, rotate for more than one week, adjust the spindle position by hand, so that the key on the spindle can enter the slot of the tool holder;
7) Record the current spindle drive display angle and set it to the drive positioning angle (refer to the drive manual);
8) The handwheel moves the Z axis up from the shank position and removes the tool magazine shank;

B. Z-axis tool change point coordinate setting

1) After adjusting the spindle positioning according to the above operation, move the Z axis to a safe position in the manual/handwheel mode, press the [spindle positioning key] to perform the spindle positioning, and then press the [tool change key] to loosen the knife, install the tool holder with a pull nail on the current cutterhead of the tool magazine to adjust the position (at this time, the tool magazine is still in the forward state and forward in place), the system switches to the handwheel mode, you can move the Z axis up and down to adjust the upward height of the Z axis of the loose knife (note that the key of the spindle in this position must be higher than the height of the tool holder tool nail in the tool magazine), record the Z-axis machine coordinates of this position, Enter the Z-axis machine coordinates into the 3rd reference point of the Z-axis (the upward point of the loose knife) [parameter number parameter #062] (generally the zero point of the Z-axis of the bucket is higher than the position of the tool magazine tool nail pulling point, when the position of the Z axis return to zero point is higher than the tool nail, the third reference point of the Z axis is set to 0, which is consistent with the 1st reference point); After setting the upward point of the loosening knife, determine that the spindle tool is in the loose state, the spindle positioning is completed, the handwheel moves up the Z axis to adjust the coordinates of the tool changing and loading point (at this time, ensure that the tool is loosened), record the adjusted Z-axis machine tool coordinates, and input the recorded Z-axis machine coordinates to the second reference point of the Z-axis (the downward loading point of the tool change) [parameter number parameter #057];
2) Press [Tool Change Key] spindle tool clamping, press [Tool magazine back key] to return the tool magazine (make sure the tool is clamped);

3) The 2/3rd reference point parameter of the Z axis must be set, otherwise the mechanical part will be damaged;

4) Turn off the tool magazine debugging function K30.7==0; Remove the knife from the spindle;

5. After passing the above two steps (point 4), you can call the tool change program O910 4 in edit mode, write the Txxx command at the beginning of the program, and then switch to automatic mode to run the program in a single section to check whether the tool change action is correct (modify the tool change program O910 4 must put the bit parameter #029.5==0, and put #029.5== after the debugging is completed 1 to avoid mistakenly modifying the tool change procedure);

6. After executing point 5, delete the Txxx command added at the beginning of the tool changer O9104, and execute the automatic tool change instruction in MDI mode after automatic mode:

Txxx M06 or M06 Txxx ;

Warning: When the spindle tool number is 0, the spindle is forbidden to load the knife, otherwise it will malfunction during automatic tool change, resulting in mechanical damage!

Row knife change macro program

Note: If the tool changer does not need to be positioned, delete the M19/M18 in the program

Choose different tool change programs according to different tool magazines

Then select the corresponding macro variable according to the tool changer

The O9104/O9107 in the system corresponds to this type of tool row and is selected according to different tool change movements

Change the tool program, and then change the program name to O9104;

#900 No. 1 knife X position	#920 11 knife X position	#940 No. 21 knife X position	#961 No. 12 knife Z position
#901 No. 1 knife Y/A position	#921 No. 11 knife Y/A position	#941 21 gauge knife Y/A position	#962 13 knife Z position
#902 No. 2 knife X position	#922 No. 12 knife X position	#942 No. 22 knife X position	#963 14 knife Z position
#903 No. 2 knife Y/A position	#923 12 size knife Y/A position	#943 22 knife Y/A position	#964 15 knife Z position
#904 No. 3 knife X position	#924 13 knife X position	#944 No. 23 knife X position	#965 16 knife Z position
#905 No. 3 knife Y/A position	#925 13 size knife Y/A position	#945 23 gauge knife Y/A position	#966 17 knife Z position
#906 No. 4 knife X position	#926 14 knife X position	#946 No. 24 knife X position	#967 18 knife Z position
#907 No. 4 knife Y/A position	#927 14 gauge knife Y/A position	#947 24 knife Y/A position	#968 19 knife Z position
#908 No. 5 knife X position	#928 15 knife X position		#969 20 knife Z position
#909 No. 5 knife Y/A	#929 15 knife Y/A position	#950 No. 1 knife Z	#970 21st knife Z

position		position	position
#910 No. 6 knife X position	#930 16 knife X position	#951 No. 2 knife Z position	#971 22 knife Z position
#911 No. 6 knife Y/A position	#931 16 size knife Y/A position	#952 No. 3 knife Z position	#972 23rd knife Z position
#912 No. 7 knife X position	#932 17 knife X position	#953 No. 4 knife Z position	#973 24 knife Z position
#913 No. 7 knife Y/A position	#933 17 knife Y/A position	#954 No. 5 knife Z position	
#914 No. 8 knife X position	#934 18 knife X position	#955 No. 6 knife Z position	
#915 No. 8 knife Y/A position	#935 18 gauge knife Y/A position	#956 No. 7 knife Z position	
#916 No. 9 knife X position	#936 19 knife X position	#957 No. 8 knife Z position	
#917 9 size knife Y/A position	#937 19 knife Y/A position	#958 No. 9 knife Z position	
#918 10 knife X position	#938 20 number knife X position	#959 10 knife Z position	
#919 No. 10 knife Y/A position	#939 20 gauge knife Y/A position	#960 11 knife Z position	

The correct way to deal with emergency stop, reset, alarm, and power failure encountered during the operation of the tool magazine:

1. In the process of automatic tool change, the tool magazine search for the command tool number is completed, the spindle has dropped to the tool loading position, and the tool has been clamped, but the tool magazine has not returned, in case of emergency stop, reset, alarm, power failure, if you need to continue to use the machine tool, you must press the [tool magazine back button] in the manual/handwheel mode when the tool magazine is returned, and then used; (Note: At this time, you cannot release the tool to move up the Z axis and then return to the tool magazine, so that the cutterhead corresponding to the spindle is equipped with a tool, leaving a hidden danger of hitting the knife for the next tool change);

2. In the process of automatic tool change, the tool magazine advances into place to the spindle to descend to the tool loading position, and the tool has been released, the Z axis has not been lifted or lifted, the cutterhead has not yet rotated, such as emergency stop, reset, alarm, power failure, if you need to continue to use the machine tool, you must observe whether the tool has been loosened in the manual/handwheel mode, if you do not release press the [tool change key] to release the tool, and then use the manual/handwheel method to lift the Z axis to a safe height, press the [tool change key] to clamp the tool, press the [tool magazine back button] to return to the tool magazine, (Note: The current spindle

corresponding to the tool magazine cutterhead is equipped with tools, there is no knife on the spindle, and the system displays T000) can be used normally;

3. In the process of automatic tool change, when the tool magazine cutterhead rotates and selects the tool, in case of emergency stop, reset, alarm, power failure, there may be inconsistencies between the current tool number and the actual tool number, the system will prompt A015.4 (abnormal tool change stop to ensure that the tool magazine is normal) alarm, if you need to continue to use the machine tool, if the tool magazine is in the forward state, press [tool change key] in manual/handwheel mode to release the tool, lift the Z axis to a safe height and press [tool change key] to clamp the tool, press [tool magazine back button] to return to the tool magazine , and reset the spindle tool number D105 (remove the tool and set it to 0), you can use it normally (do not set it when the current tool number is 0).

Appendix IV: Use of Tool Instruments

Relevant signals

Signal	Tool setting instrument signal	Tool setter overtravel signal
Diagnostic address	X0.4	X1.7
Interface pins	CN61.05	CN61. 16
remark		

Relevant parameters:

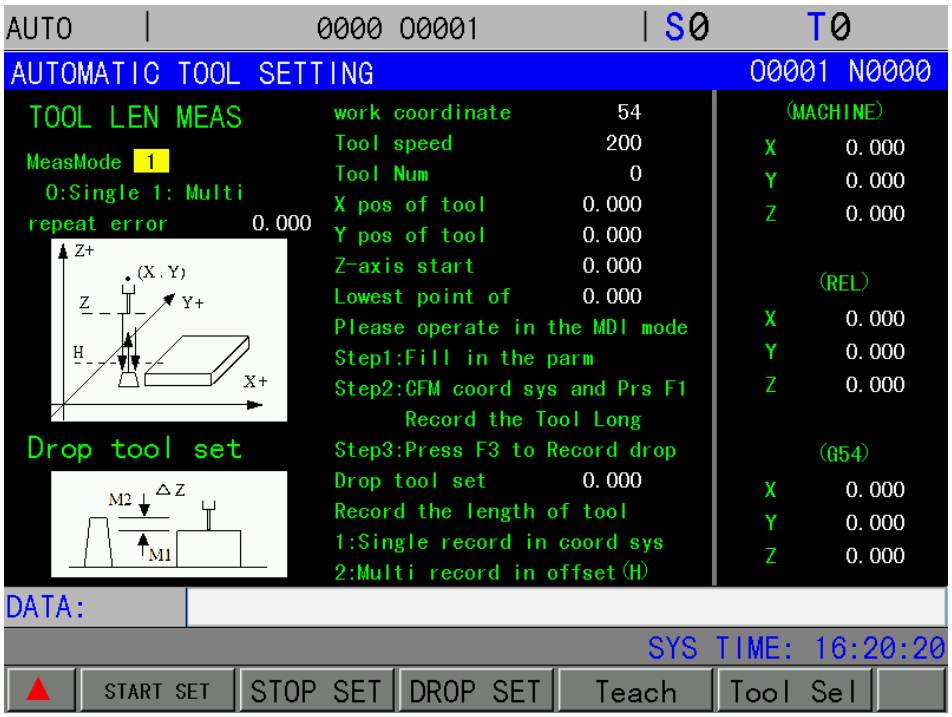
PLC parameters:

K020.3==1 Key automatic tool setting function (0: invalid 1: valid).

K024.6==0 The tool setter overtravel signal is (0:normally closed 1: normally open) valid (normally closed contact).

K024.7==0 The input signal of the tool setter is (0: normally closed 1: normally open) active (normally closed contact).

Tool setting interface



Parameter description:

1. Measurement mode: 0 single-knife mode 1 multi-knife mode

When selecting the single tool mode, the knife directly changes the coordinate value of the Z axis of the coordinate system, which is used with the workpiece coordinate system; The drop setting value of the single-tool mode is reflected in the Z-axis position of the base offset, so when the tool is not changed but only the workpiece is changed, the drop can be measured directly.

When selecting the multi-tool mode, change the tool length to compensate H value, and compensate for the tool offset number according to the current tool decision; The drop setting value of the multi-tool mode will be reflected in the Z-axis position of the selected workpiece coordinate system, so when the tool is not changed but only the workpiece is changed, the drop can be measured directly.

2. Repeated tool setting error value:

When the data difference between the two knife sets exceeds the set error value, the system alarm prompts the knife to exceed the error, but the system will record the data for the second time, whether to use it is determined by the customer, if the data is used is to press the reset to cancel the alarm; When the second tool setting data is not in use, press reset to cancel the alarm and reset the tool again.

3. Workpiece coordinate system

Select the standard system you want to set the tool, when using the G54 coordinate system, you need to fill in 54; When using the G55 coordinate system, you need to fill in 55; And so on until the G59 coordinate system.

4. Knife speed:

The tool moves quickly above the tool setter and touches the speed of the tool setter at the speed of the second segment;

(When touching the tool setter, raise the Z axis up 5 mm, and then move down 6mm at the speed of F 50, if the tool setter system alarm; If the system records the tool setting data when the tool setting instrument is encountered, raise it by 5 mm, and then move it down by 6mm at the speed of F 50, if the tool setting instrument system alarm is not encountered; If the tool setting instrument is encountered, the system records the tool setting data, compares the two tool setting data values, and alarms if the set error value is exceeded).

5. The current knife number

When selecting multiple knives, you can directly fill in the knife number that needs to be set, and automatically exchange the required knife number when starting the knife setting; This parameter changes automatically when a tool is selected in the tool selection screen.

6. X-axis coordinates of the position of the tool setter

Y-axis coordinates of the tool setting instrument position

This coordinate value is directly above the tool setter and can be taught by the teach key.

7. Z coordinate of the starting point of the tool

Quickly move to the Z-axis position above the tool setter, and the coordinate value can be taught by the teaching key; The tool quickly moves to this coordinate and then touches the tool setter at the tool setting speed. When setting this coordinate, you should consider whether there is a collision tool setter problem when installing a long tool.

8. The lowest point of the **Z** axis **of the tool setting instrument**

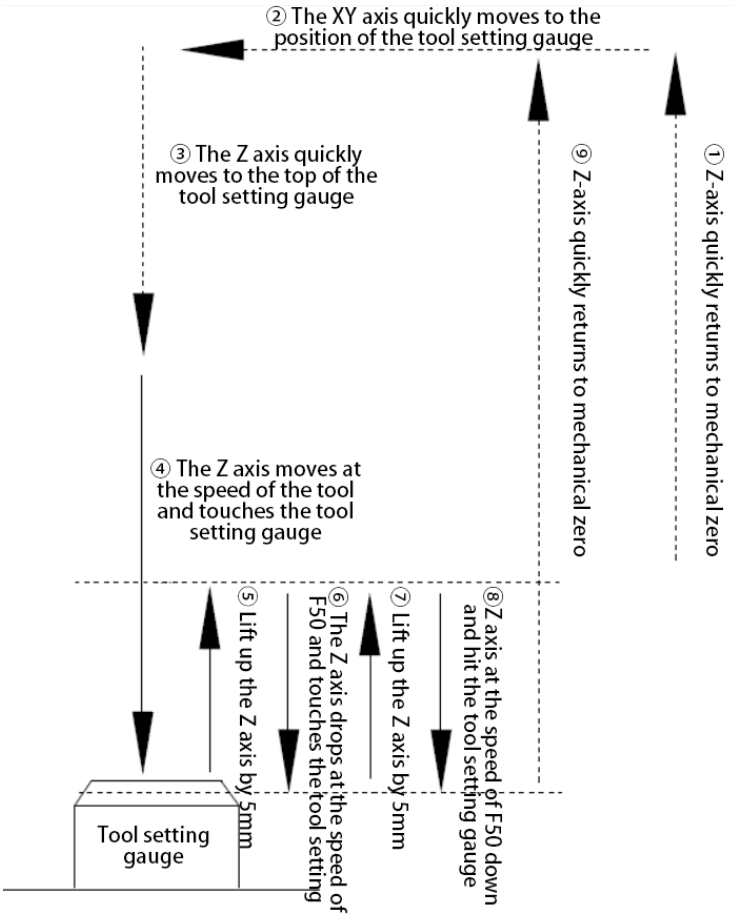
The Z-axis can be moved to the lowest mechanical coordinate value when setting the tool, and the coordinate value can be taught by the teaching key.

When setting this coordinate, you should consider whether the tool is set correctly when installing a short tool.

9. Z-axis drop setting

This data is measured without manual input; After the tool setting instrument is completed, the handwheel moves the tool to the origin of the Z axis of the workpiece, presses [drop setting], and the system automatically calculates the height difference between the origin of the Z axis of the workpiece and the tool setter.

Knife action steps:



Automatic tool setting operation: single tool mode, taking the G54 coordinate system as an example:

The parameter setting of the tool setting instrument is completed; Enter the automatic tool setting interface in MDI mode, press the tool setting to start, the tool automatically runs the tool setting, after the completion of the tool setting instrument data is recorded in the Z axis of the G54 coordinate system, switch to the handwheel mode to move the Z axis to the workpiece Z axis original click the [drop setting] button, The system calculates the drop value between the workpiece origin and the tool setter, and records the Z-axis coordinates of the base offset; At this time, the Z-axis tool setting is completed, and the XY axis G54 coordinates can be switched to the coordinate system interface.

When the tool is replaced and the workpiece is not replaced, it is only necessary to re-align the Z-axis tool length without resetting the drop;

When the tool is not replaced and the workpiece is replaced, it is only necessary to reset the drop without re-aligning the tool length.

If the Z axis uses the tool holder rolling method to the workpiece surface, after setting the drop, the drop value is recorded in the Z axis base offset, at this time, you need to add the tool holder diameter value to the Z axis offset, and the operation step → the cursor to move to the Z axis base offset position→ enter the tool holder diameter value→ press [Modify] or increment input to → the tool pairing, then the Z offset value = the tool setting instrument drop value + tool holder diameter.

Automatic tool setting operation: multi-tool mode, taking the G54 coordinate system as an example:

The parameter setting of the tool setting instrument is completed; Enter the automatic tool setting interface in MDI mode, select the tool number that needs to be paired on the tool selection interface, or fill in the tool

number that needs to be taken at the current tool number, and the tool alignment instrument data is recorded in the tool offset number corresponding to each tool number; After the tool length alignment is completed, select one of the knives, press the [drop setting] key when moving the tool to the origin of the Z axis of the workpiece with the handwheel, and the system calculates the drop value between the origin of the workpiece and the tool setting instrument, which is recorded in the selected workpiece coordinate system G54 Z axis coordinates; Switch to the coordinate system interface to take the XY axis G54 coordinates.

When the tool is replaced and the workpiece is not replaced, it is only necessary to re-align the Z-axis tool length without resetting the drop;

When the tool is not replaced and the workpiece is replaced, it is only necessary to reset the drop without re-aligning the tool length.

If the Z axis uses the tool holder rolling method to the workpiece surface, after setting the drop, the drop value is recorded in the coordinate system of selecting the tool pairing, at this time, you need to enter the tool holder diameter value in the Z axis, and the operation step → the cursor moves to the Z axis base offset position→ enter the tool holder diameter value→ press [Enter] or [Enter] or absolutely enter → to finish the tool pairing, then the Z offset value = the diameter of the tool holder.

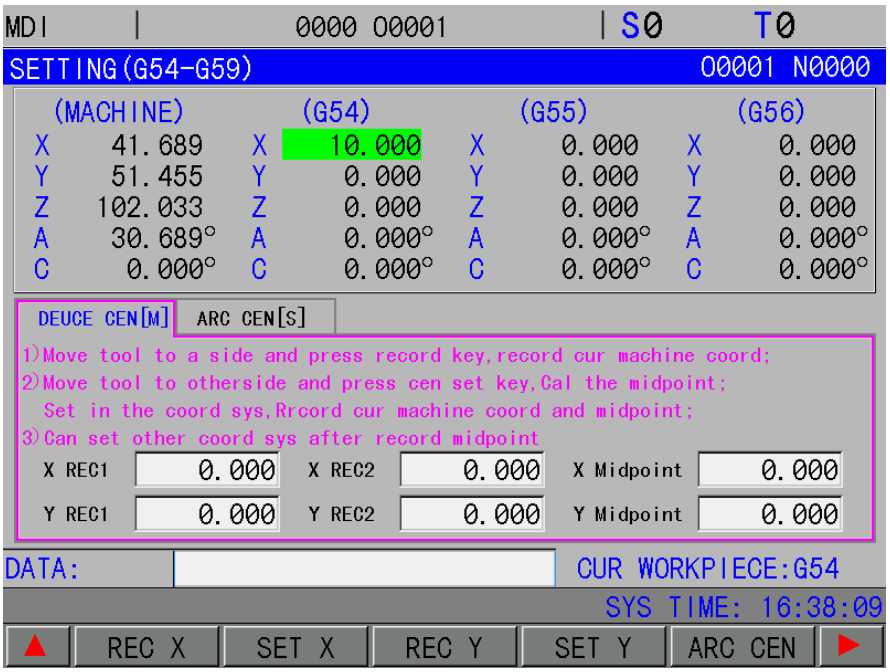
For example, T01 knife is recorded in knife No. 01, and knife TO2 is recorded in knife No. 02; Use with G43 H_; For example:

M06 T01;

G54;

G43 H01 Z20;

Appendix V: Use of Automatic Classification Function



In two points: (XY axis).

Press [Settings] to enter the setting interface and then press [Coordinate System] to enter the sub-center interface, use the handwheel or manually move the X axis to one end of the workpiece X axis direction and press [Record X] and then move the X axis to the other end of the workpiece and press [Minute X], the workpiece X axis direction is completed with the tool, and the center coordinates are automatically input into the workpiece coordinate system where the light standard is located.

Use the handwheel or manually move Y to one end of the workpiece Y direction and press [Record Y], and then move Y to the other end of the workpiece and press [Split Y] workpiece Y direction to complete the tool setting, and the center coordinates are automatically entered into the workpiece coordinate system where the light standard is located.

Circular arc minutes: (XY axis, three points in the middle).

Press [Settings] to enter the setting interface and then press [Coordinate System] to enter the sub-middle interface

Use the handwheel or manually move the XY axis to the arc workpiece on the previous click [Record 1] and then move the XY axis to the workpiece to another click [Record 2], and finally move the XY axis to another point of the workpiece, press [Minute], the arc is completed against the tool, and the center coordinates are automatically entered into the workpiece coordinate system where the light standard is located.

Appendix VI: CAD Drawing DXF File Import Drilling Tapping

Function description: The system can set drilling and tapping by recognizing the hole size of the DXF file plan in the U disk (solid or dashed line can be read), and the process selection can choose to punch points (drill center holes), drilling, tapping, chamfering, and generate program processing order of dotting → drilling → chamfering→ tapping; In each selection, there is a default processing data, which users fill in and modify according to their own needs.

Related parameters:

Parameters: 31.1 CAD (DXF file) punching and tapping function (0:No 1:Yes) is valid

31.2 CAD Generator Tool Change Instruction M 06 T_ Add M00 instruction under (0:Yes 1:No); (The need to open function that requires manual tool change in the absence of a tool magazine, manual tool change when performing a pause until M00).

31.4 CAD default tool number (0:1 tool 1:100 tool) (tool T100 is generated if the default 100 tool machining parameters are not modified.) When the system program alarms illegal compensation number).

31.5 When sorting dots, the less moving axis starts from the coordinates of the numeric value (0:small 1:large) (default parameters can be used to set the dotting action order).

31.6 When sorting dots, the multi-moving axis starts from the coordinates of the numeric value (0:small 1:large) (the default parameters can be used to set the order of dotting actions).

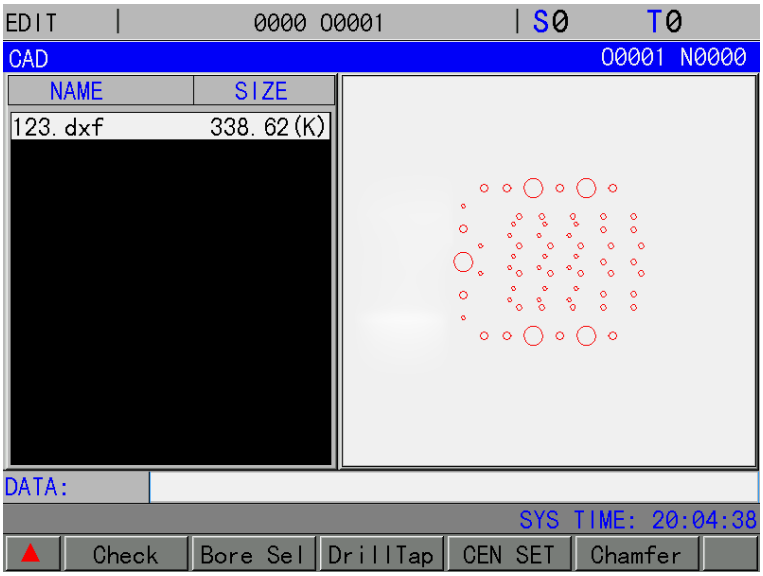
31.7 The multi-moving axis is (0:Y 1:X) axis when sorting dots (default parameters can be used to set the

dotting action order).

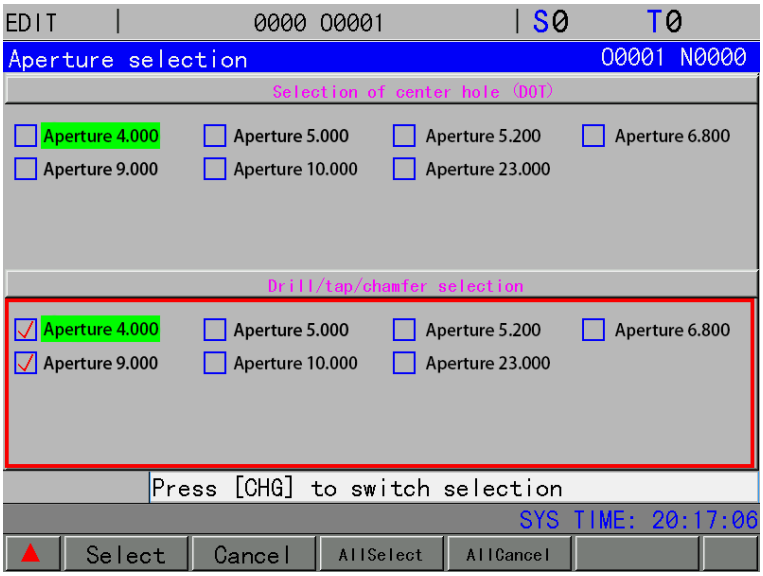
How to enter the CAD import interface:

1) Need to turn on the CAD hole tapping function is effective, and then restart;
2) Insert the U disk with CAD (DXF format) files in the root directory into the system U disk interface;
3) Press [Program Key] to enter the program interface, press [] to move the menu to the right to find the [▸ CAD function] menu;
4) Press the function button [F1~F5] corresponding to [CAD function] to enter the CAD function;
5) Select the required DXF file in the file name box and press the [View confirmation] function button [F1~F5] to view CAD diagram;
6) After checking and confirming, press the [Aperture Selection] corresponding function button [F1~F5] to check the aperture that needs to be punched or dotted;
7) Press the corresponding function button [F1] for [Drilling and tapping], [Point setting], and [Chamfer setting]. ~F5】Set processing data;
8) Finally, press [Generate program] on [Drilling and tapping] or [Dotting settings] or [Chamfer settings].] Corresponding function button [F1~F5] to generate the processing program, the processing program is found in the local directory (set the generated program name).

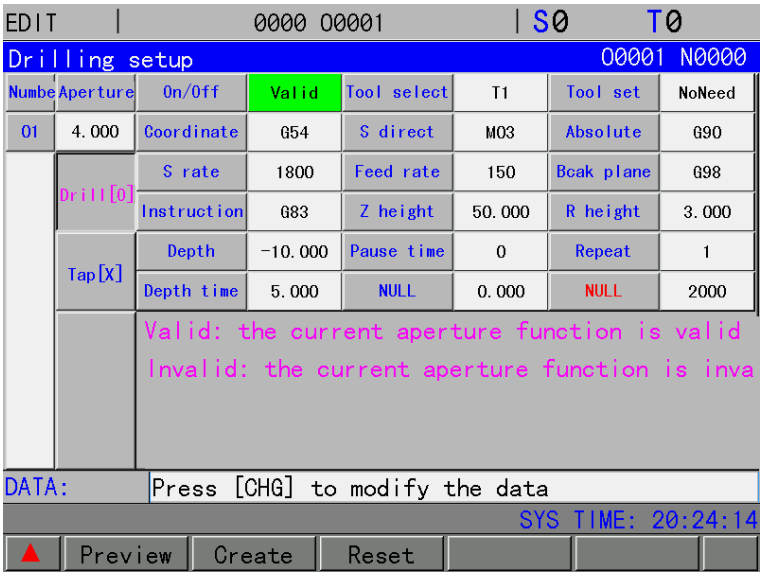
The interface displays:



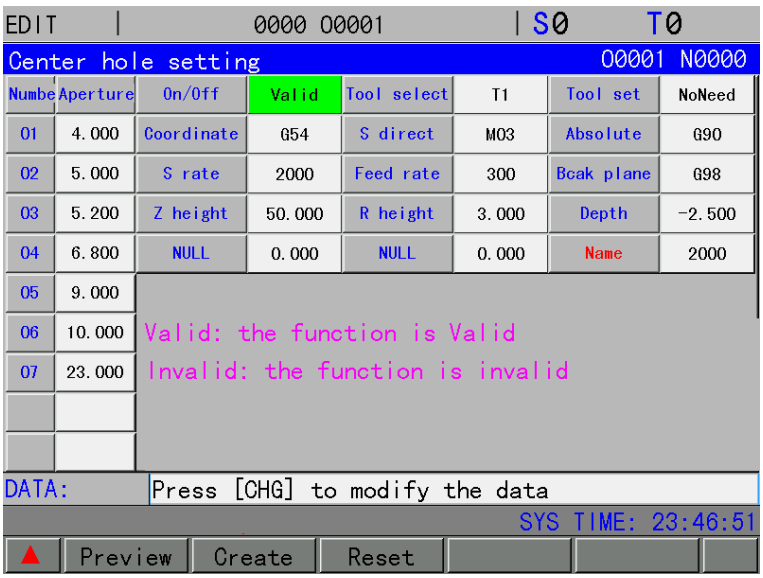
[View confirmation]. The interface is displayed



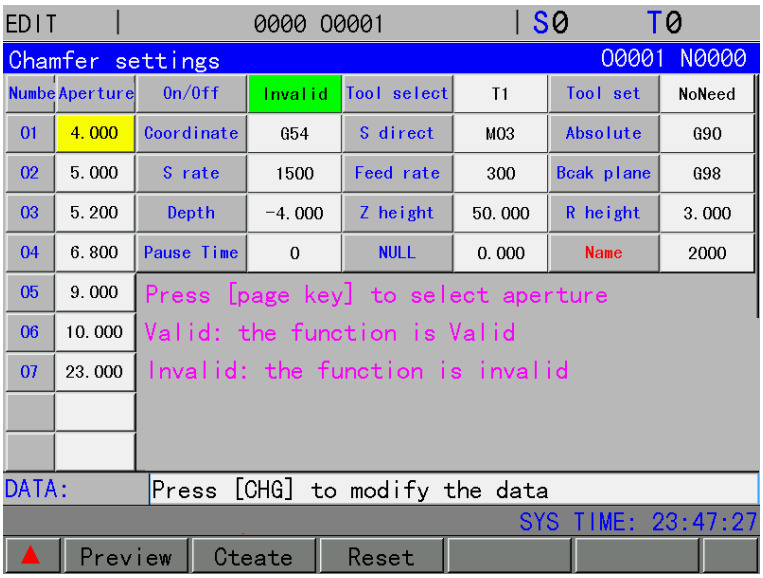
[Aperture selection]. The interface is displayed



[Drilling and tapping]. The interface is displayed



[Dot settings]. The interface is displayed



[Chamfer setting]. The interface is displayed

Appendix VII Macro Program Supplements

1. Statement-based macro code (macro B).

1.1 Arithmetic and logical operations

Arithmetic and logical operations

function	Expression structure	remark
Define or assign a value	#i=#j	
addition subtraction multiplication division	#i=#j+#k #i=#j-#k #i=#j*#k #i=#j/#k	
or and XOR	#i=#j OR #K #i=#j AND #K #i=#j XOR #K	Logical operations are performed bit by bit by binary number
square root absolute value rounding Round up Round down Napierian logarithm Exponential function	#i=SQRT[#j] #i=ABS[#j] #i=ROUND[#j] #i=FUP[#j] #i=FIX[#j] #i=LN[#j] #i=EXP[#j]	
sine arcsine cosine inverse cosine tangent Anyway tangent	#i=SIN[#j] #i=ASIN[#j] #i=COS[#j] #i=ACOS[#j] #i=TAN[#j] #i=ATAN[#j]/[#k]	The unit of angle is specified in degrees, e.g. 90°30' is expressed in 90.5 degrees
Switching from BCD to BIN Switching from BIN to BCD	#i=BIN[#j] #i=BCD[#j]	Used for conversion with PMC signals

Related instructions

1、Angular units

The angular units of the functions SIN, COS, ASIN, ACOS, TAN, and ATAN are degrees (°). For example, 90°30′ should be expressed as 90.5° (degrees).

2、Arcsine #i=ASIN[#j]

i、The range of the result output is as follows,

When the bit parameter 25.7 NAT bit is set to 1, 90°~ 270°;

When the bit parameter 25.7 NAT bit is set to 0, -90°~ 90°;

ii、When #j goes outside the range of -1 to 1, a P/S alarm is raised.

iii、Constants can replace the variable #j.

3、Anticosine #i=ACOS[#j]

i、The result output range is from 180°~0°;

ii、When #j exceeds the range of -1 to 1 P/S alarm;

iii、Constants can replace the variable #j.

4、Anywaytangent #i=ATAN[#j]/[#k]

Specify the length of both sides, separated by a slash '/'.

i、The value range is as follows:

When the bit parameter 25.7 NAT bit is set to 1: 90°~270°;

[Example] when specifying #1=ATAN[-1]/[-1] #1=225°;

When the bit parameter 25.7 NAT bit is set to 0 -90°~90°;

[Example] when specifying #1=ATAN[-1]/[-1] #1=45.0°;

ii、Constants can replace the variable #j.

5、Natural logarithm #i=LN[#j]

i. Constants can replace the variable #j.

6、Exponential function #i=EXP[#j]

i、Constants can replace the variable #j.

7、ROUND rounding function

When the ROUND function is included in the arithmetic or logical operation code IF or WHILE, the ROUND function is rounded at the first decimal place.

For example:

When #1=ROUND[#2] is executed, at this point #2=1.2345, the value of variable 1 is 1.0.

8、Round up and round down

When CNC processes numerical operations, if the absolute value of the integer generated after the operation is greater than the absolute value of the original number, it is called rounding; If it is less than the absolute value of the original number, it is called rounding. Care should be taken with the handling of negative numbers.

For example:

Suppose #1=1.2 and #2=-1.2

When #3=FUP[#1] is executed, 2.0 is assigned to #3.

When #3=FIX[#1] is executed, 1.0 is assigned to #3.

When #3=FUP[#2] is executed, -2.0 is assigned to #3.

When #=FIX[#2] is executed, -1.0 is assigned to #3.

1.2 Transfer and Circulation

In the program, the use of GOTO statements and IF statements can change the flow direction of control. Three transfer and loop operations are available.

1、GOTO statements (unconditional transfer).

2、The condition controls the IF statement.

3、WHILE loop statement.

1）Unconditional transfer (GOTO statement)

The program segment transferred to the sequential number n. When an alarm is assigned a sequence number other than 1 to 99999, you can specify the sequence number using an expression.

Format: GOTOn; n: Sequential number (1~99999).

Example: GOTO1;

GOTO#101;

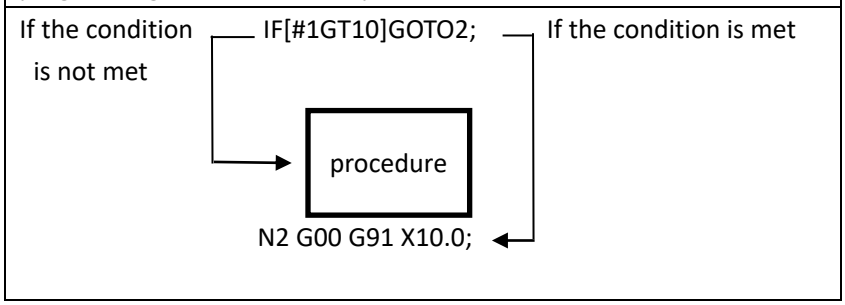
2） Conditional Control (IF Statement)

GOTO format: IF [conditional expression] GOTOn;

If the specified conditional expression is true, transfer to the program segment with the sequential number n; If the specified conditional expression does not hold, the next program segment is executed sequentially.

Example:

If the value of variable #1 is greater than 10, transfer to the program segment with the sequence number N2.



THEN format: IF [conditional expression]THEN< macro program statement >;

If the conditional expression is true, only one statement can be executed after THEN.

Example: IF[#1EQ#2]THEN#3=0;

If the value of #1 is equal to the value of #2, assign 0 to variable #3; if not, the assignment statement after THEN is executed in order.

Conditional expressions: Conditional expressions must include conditional operators, which can be variables, constants, or expressions on both sides, enclosed in parentheses '['']'.

Conditional operators: The conditional operators listed in the following table are available for the system.

Conditional operators	meaning
EQ or ==	Equals (=)
NE or <>	Not equal to (≠).
GT or >	Greater than (>).
GE or >=	Greater than or equal to (≥).
LT or <	Less than (<).
LE or <=	Less than or equal to (≤).

For example: IF[3<>2]GOTO2; Its meaning is: if 3 is not equal to 2, jump to the N2 program segment;

IF[#101>=7.22]THEN#101=SIN30； Its meaning is: if #101 is greater than or equal to 7.22, then the assignment operation after THEN is performed. That is, a sine of 30 degrees is assigned to variable #101.

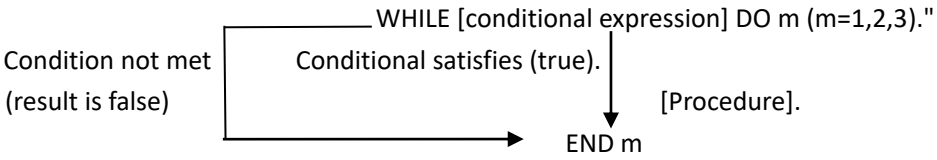
Typical program: The following program calculates the sum of integers 1~10.

```
O9500
#1=0；..... and initialized to 0
#2=1；..... The initial value of the added number is 1
N1 IF[#2 GT 10]GOTO2；..... When the added number is greater than 10, it is transferred to N2
#1=#1+#2；..... Calculate the sum of two numbers
#2=#2+1；..... Be added to add 1
GOTO1； ..... Unconditionally jump to program segment N1
N2 M30； ..... End of procedure
```

3）**LOOP (WHILE STATEMENT)**

Specify a conditional expression after WHILE, and when the specified condition is true, execute the program segment from DO to END; Otherwise, jump to the program segment after END.

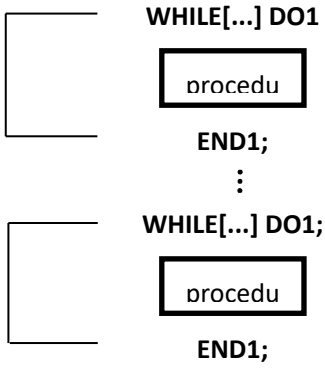
Example:



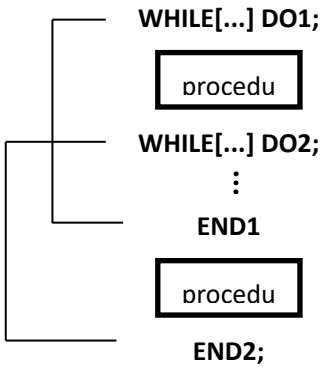
Note: When the specified conditions are true, execute the program segment from DO to END; Otherwise, execute the program segment after END instead. The label after DO and the label after END should be consistent, and the label value can be 1, 2 or 3.

Nesting: DO, the label (1~3) in the END loop can be used as many times as needed. However, an alarm is issued when there is a cross-repeating loop in the program.

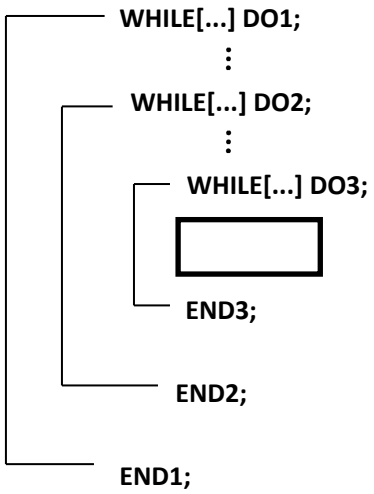
1. Labels (1 to 3) can be used multiple times upon request



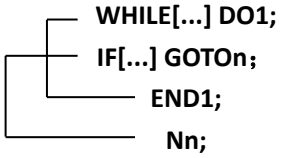
2. **The ranges of DO cannot be crossed**



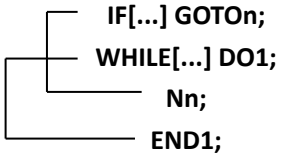
3. **DO loops can be nested 3 levels**



4. **The control can go to the outside of the loop**



5. **Transfers cannot enter the circulation zone**



1.3 Macro alarm

Format: ERR_;
Range 0~200;
Alarm number 3000+n;
For example: E RR01 or E RR1;
The system alarm number is 3001.

1.4 Special instructions (substitution of variable numbers with variables).

Format: "#" + "9" + permutation variable number
Example: #100=205, #205=500;
X#9100 and X500 codes function the same;
The X-#9100 and X-500 codes function the same;

It can be understood as #9_ _ _, followed by 3 digits as a common variable number, in the range 100~199,500～999;

#9100 represents the macro variable number represented by the value within #100;

For example, #100=510, #510=101;

At this time, write #9100, #9100=#510=101;

Appendix VIII: Preparation of custom alarm content

Bit parameter 32.3 [Read macro alarm/comment (0:no 1:Yes) valid]

Macro A alarm

G65H99P_;//[ERR_, the system displays the alarm content] (within 25 words).

Range 0-200;

Macro B alarm

ERR_;//[ERR_, the system displays the alarm content] (within 25 words).

Range 0-200;

For example:

ERR1;//[ERR1, quantity arrived];

The system displays the macro program alarm 3001 quantity arrived;

Note: At the same time, the value of the system macro variable #3000 can be written for alarm, the range is also 0-200, and the alarm number is also consistent with 3000-3200; The writing method is G65 H01 P#3000 Q_, or #3000= _;

Example: G65 H01 P#3000 Q0;

#3000=0;

G65H99P0;

ERR0; These four weaves have the same effect.

Modify the system to read the program alarm comments, then display the alarm content and report it in the system file WARN2;

Program macrovariable comments can be read and saved in the system file MacroVar;

Specify the scope of the writeable alarm;

0650-0657； M89K0-K7 detection

3000-3200； Macro program alarm P0-P200

；

Write M89K0-K7 alarm specification format:

M89 K0-K7 J_ ;//[ERR650-657,M89K0-K7 detection alarm];

M89K0-K7 detection alarm; The alarm display content is recorded in the file WARN, and if the alarm content is not written when the same alarm number is present, the stored alarm display is directly read.

For example:

M89K3J5000;//[ERR653, M89K3 jacking not in place]

or

[ERR653, M89K3 jacking not in place]

M89K3J5000;

As long as the program reads [ERR653, M89K3 top material is not in place], the alarm content is saved in the system file 6No. 53 alarm;

When the next time the alarm display is not written, the same code is instructed, and the same alarm will be displayed before the content written;

For example:

M89K3J5000;//

At this time, the alarm will still show that the M89K3 top material is not in place;

Write macro program alarm specification format

Macro A:

G65H99P_;//[ERR_, alarm display content];

Macro B:

ERR_;//[ERR_, alarm display content];

The alarm display content is recorded in the file WARN2, and if the alarm content is not written when the same alarm number is present, the stored alarm display is directly read.

For example:

G65H99P0;//[ERR0, processing quantity arrived].

or

[ERR0, the number of processing arrived].

G65H99P0;

As soon as the program reads [ERR0, processing quantity arrives], the alarm content is saved in the system file alarm number 3000;

When the next time the alarm display is not written, the same code is instructed, and the same alarm will be displayed before the content written;

For example:

G65H99P0;

or

ERR0;

At this time, the alarm will still show the macro alarm 3000 processing quantity arrived;

The program reads macro variable comments saved in the system file MacroVar:

Comments can be changed:

#100-#199；

#500-#999； (#900-#999 reserved for internal use of the system);

Modify the annotation specification format;

//[#--, display content] (20 words or less).

//[#--, display content]

The display content will be saved in **MacroVar and can be viewed in a public variable;**

Macro variable comments are saved without changing the same macro variable comment.

Appendix IX, M-Code Table

M code	definition	remark
M00	Unconditional suspension	The program pauses when running to M00
M01	Select Pause Stop	The program pauses when running to M01 and the system has the Select Stop function turned on, otherwise it will run down
M02	End of procedure	
M03	The spindle is spinning	Interlock, boot default M05
M04	The spindle is reversed	
M05	The spindle stops	
M06	Automatic tool change	
M07	Processing blowing	M07 and M 09 interlock, M 08 and M 09 interlock, boot default M 09
M08	Coolant	
M09	Turn off process blowing/cooling	
M10	Table clamping	Interlock
M11	The workbench is released	
M16	The spindle tool is released	Interlock, boot default M17
M17	Spindle tool clamping	
M18	Spindle positioning canceled	Interlock
M19	Spindle positioning output	
M20	retain	All reserved M codes are invalid function codes
M21	Tool magazine forward/pocket vertical	It cannot be used alone, it needs to be used with the tool magazine, written in the tool change program, and can only be used normally after M 50.
M22	Tool magazine finder	
M23	Arm clasp knife	
M24	Tool magazine setback/pocket level	
M25	The knife arm rotates	
M26	The knife arm returns to the origin	
M27	retain	All reserved M codes are invalid function codes
M28	Rigid tapping canceled	Interlock
M29	Rigid tapping opens	
M30	End of procedure	
M31	The number of processed pieces is added to 1	
M32	Lubricating output	Interlock
M33	Lubrication is turned off	
M34	Guard door output	Interlock
M35	The protective door is closed	
M36	Chip conveyor output	Interlock

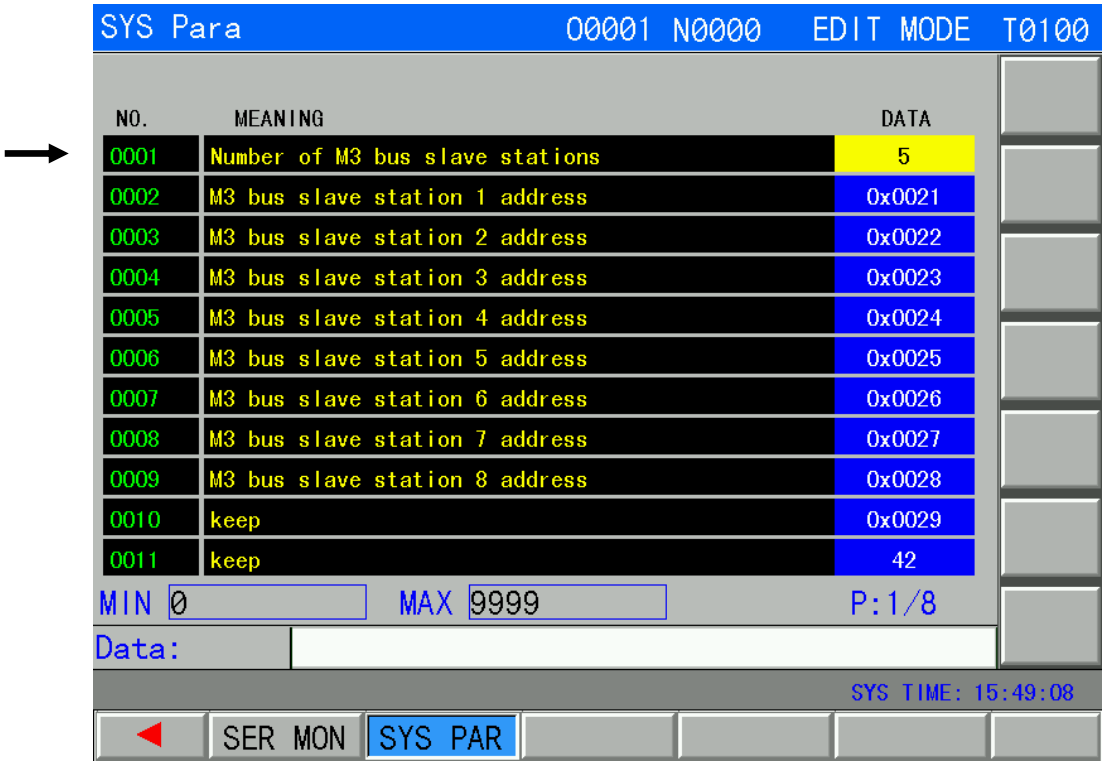
M37	The chip conveyor is turned off	
M38	Work light output	Interlock
M39	The work light is off	
M40	retain	All reserved M codes are invalid function codes
M41	The spindle shifts 1 gear	Interlock
M42	The spindle shifts 2 gears	
M43	The spindle shifts 3 gears	
M44	The spindle shifts 4 gears	
M45	retain	All reserved M codes are invalid function codes
M46		
M47		
M48		
M49		
M50	The knife change begins	Interlock
M51	End of knife change	
M52	retain	All reserved M codes are invalid function codes
M53		
M54		
M55		
M56		
M57		
M58		
M59		
M60		
M61		
M62		
M63	The second spindle rotates forward	Interlock, boot default M65
M64	The second spindle is reversed	
M65	The second spindle stops	
M66	retain	All reserved M codes are invalid function codes
M67		
M68		
M69		
M70	Blow on the knife output	Interlock
M71	Turn off the knife by blowing air	
M72	CustomizeK2 output	Interlocked, lead screw clamped
M73	CustomK2 off	
M74	CustomizeK3 output	Interlock
M75	CustomK3 off	
M76	retain	All reserved M codes are invalid function codes

M77		
M78	Restore feed rate 100%.	Interlock (automatic tool alignment using ladder diagram version V3.20 or higher).
M79	Restore the original feed rate	
M80	retain	All reserved M codes are invalid function codes
M81		
M82		
M83		
M84		
M85		
M86		
M87		
M88		
M89	Customize the output	Multi-format programming
M90	The program segment repeats to end	Interlock, M 91 C_ needs to end with M90
M91 C	The program segment starts repeatedly	
M92	Conditional jumps	
M93	Unconditional jumps	
M94	retain	All reserved M codes are invalid function codes
M95		
M96		
M97		
M98	Call the subroutine	If you do not write L times, it is called once by default
M99	End of subroutine	When written in the main program, the program is an infinite loop
M9000—M9999		A single call to the O 9000-O9999 program

Appendix 10, MIII. Bus System Connection Settings (available for bus version).

Note: MIII. bus version system, there is [bus configuration] menu to set bus connection parameters, only need to set the number of communication axes and communication address and gear ratio;

1) First go to the [System Parameters] interface:

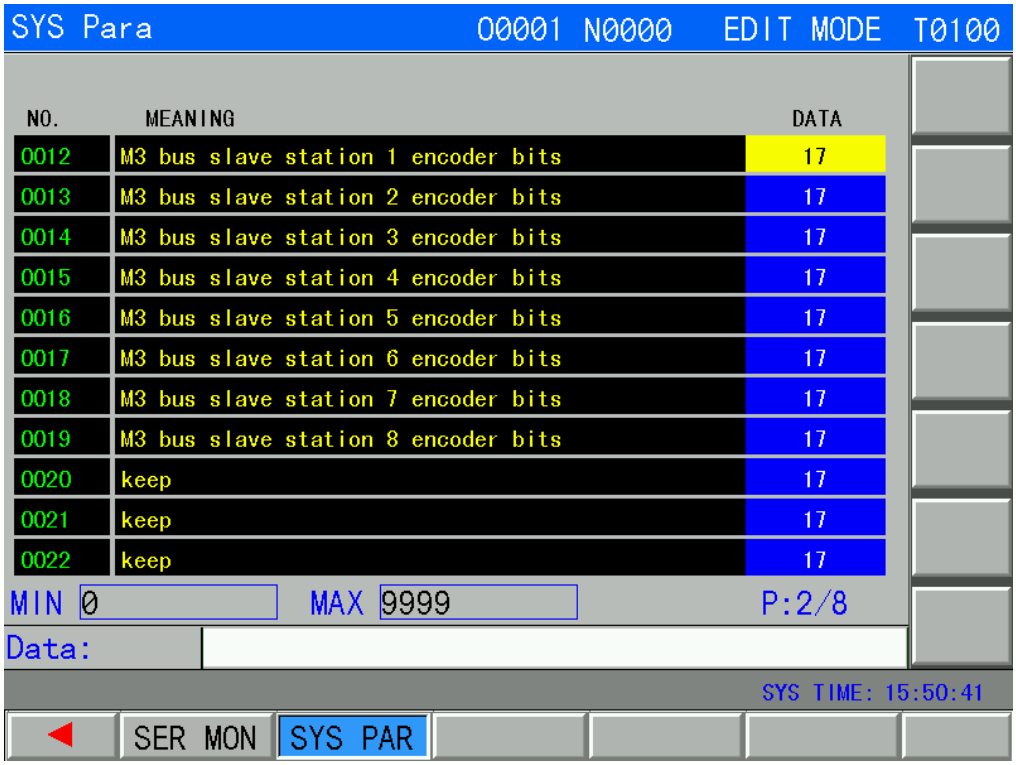


Note 1: 001 is the number of communication slaves, when there are two bus axes, it is set to 2; At this time, the system reads the addresses of slave address 1 and 003 02 and slave address; The slave address needs to correspond to the drive communication address;

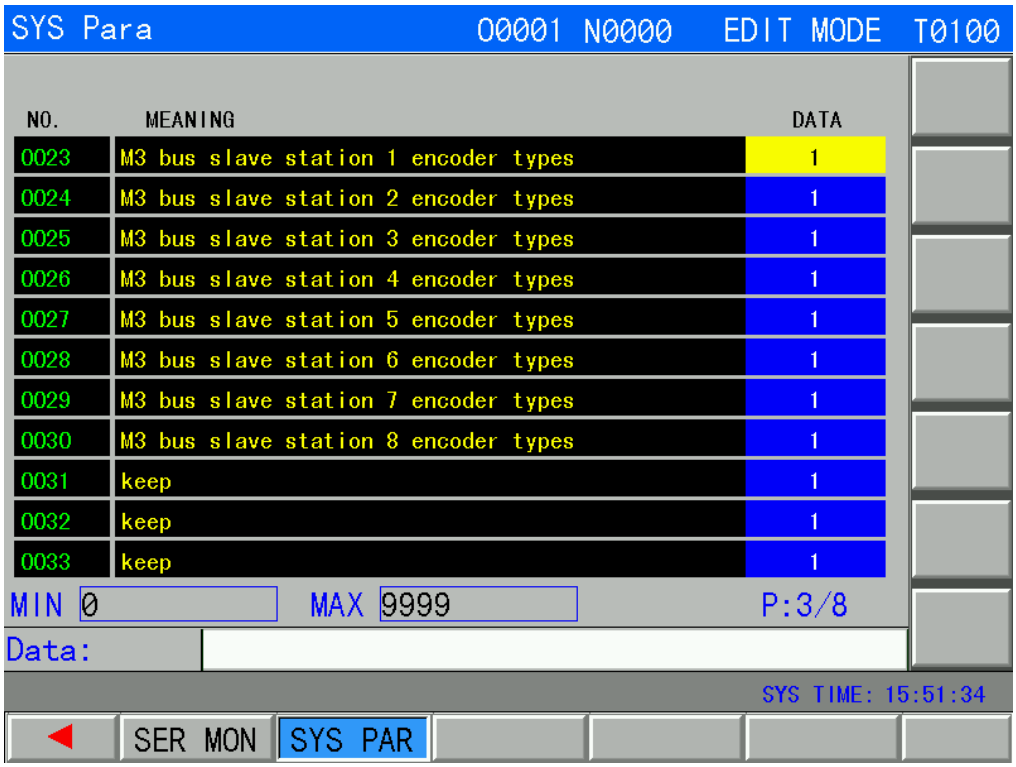
Note 2: The axes corresponding to the number of slaves are: slave 1 corresponds to the X axis, slave 2 corresponds to the Y axis, slave 3 corresponds to the Z axis, and the slave station 4 corresponds to 4 TH axis, slave 5 corresponds to 5TH axis;

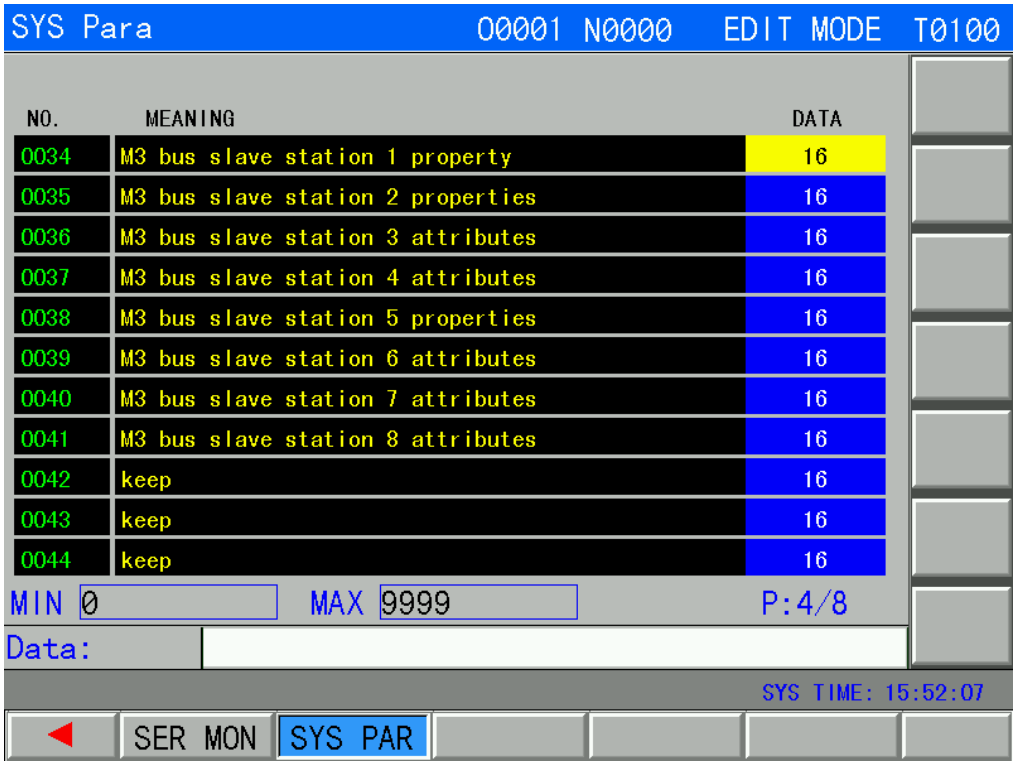
Note 3: The slave station address starts with 2 1, corresponding to Yaskawa drive communication address 21; When the KY300AWM3 driver is connected, the slave address 2 1 corresponds to the drive communication address 1, and the slave address 2 2 corresponds to the drive 2 , slave address 2 3 corresponds to drive 3, and so on;

Note 4: The remaining reserved parameters do not need to be set.

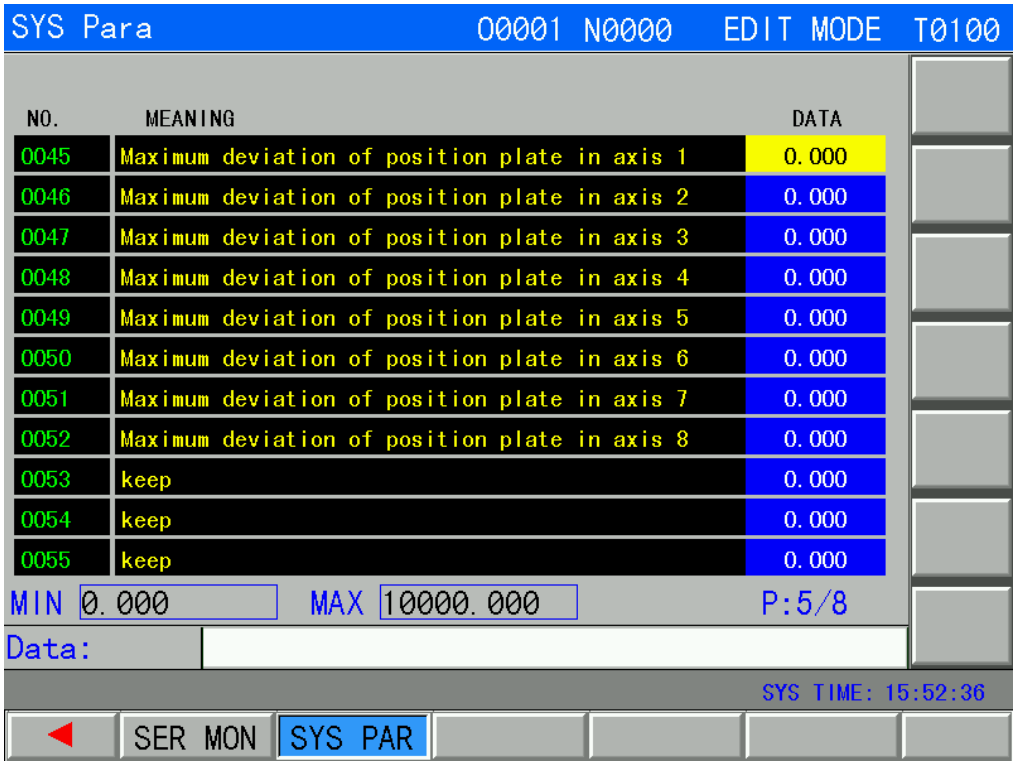


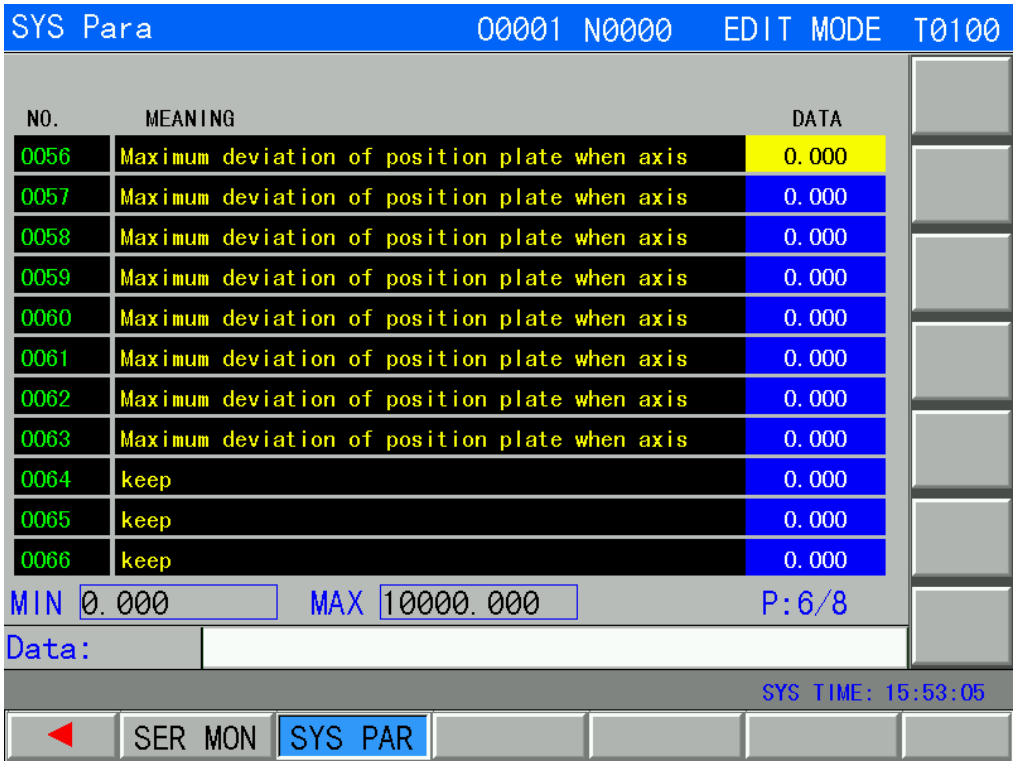
Note: The encoder number parameter is 17 by default and does not need to be set.



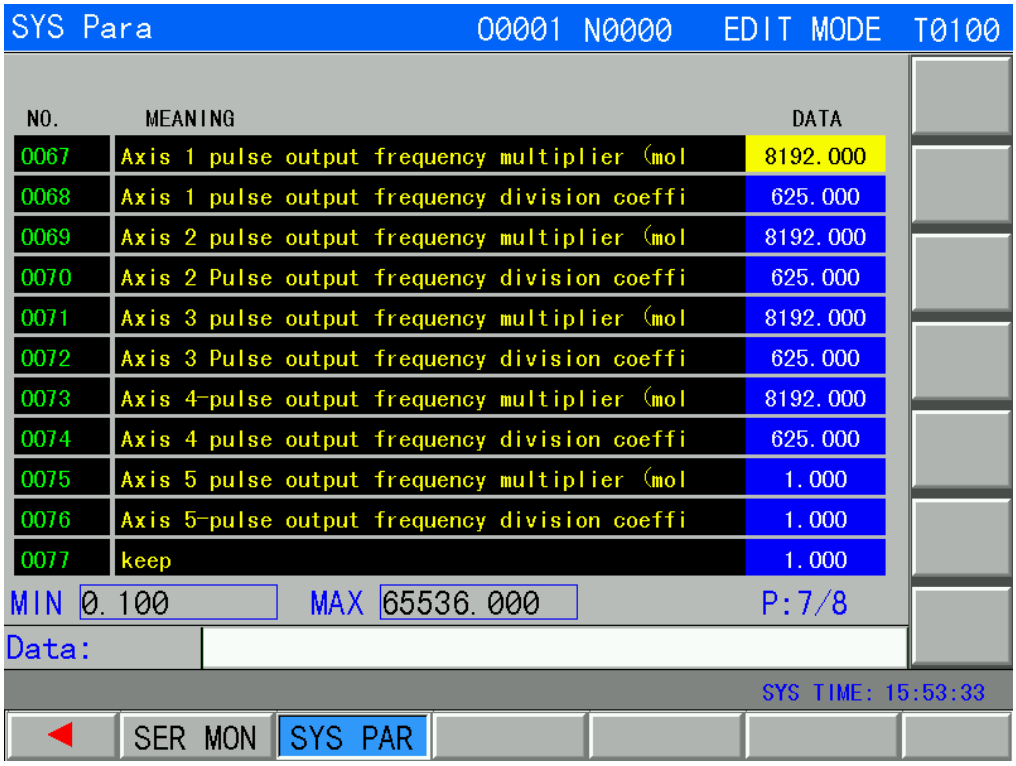


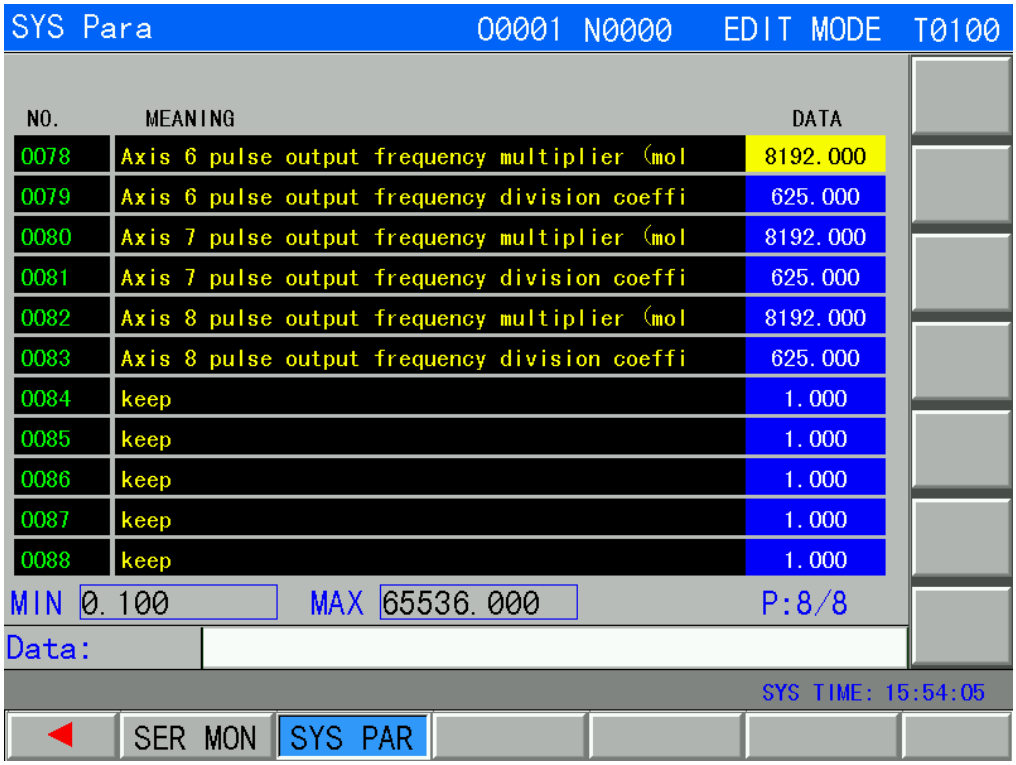
Note: Neither the encoder type nor the slave properties need to be set (if the motor encoder is incremental, the encoder type needs to be set to 0).





Note: Motion and stop deviations cannot be set at this time.





Note: The bus configuration default motor encoder number of 1 7 bits, the default gear ratio 8192/625, this gear ratio is the default counting gear ratio, the motor is 17 bits or 2 3 None of the bits need to modify this parameter value;

For those using spindle pulse speed, it is necessary to turn on the 6th axis for the analog spindle function [bit parameter 7.0 6th axis (0: No 1: Yes) set to analog spindle], gear ratio set to parameter 95/ The numerator is 1, [9 6]. With a denominator of 36, the system generates 1 0,000 pulses per revolution (the corresponding shaftbus configuration gear ratio needs to be set to 1:1).

Note: When using the M3 bus spindle servo, it is necessary to set the corresponding axis to the pulse spindle function, and it needs to be set at the same time

bit parameter 51.4 [M3 communication last address as bus spindle function (0: invalid 1: valid)];

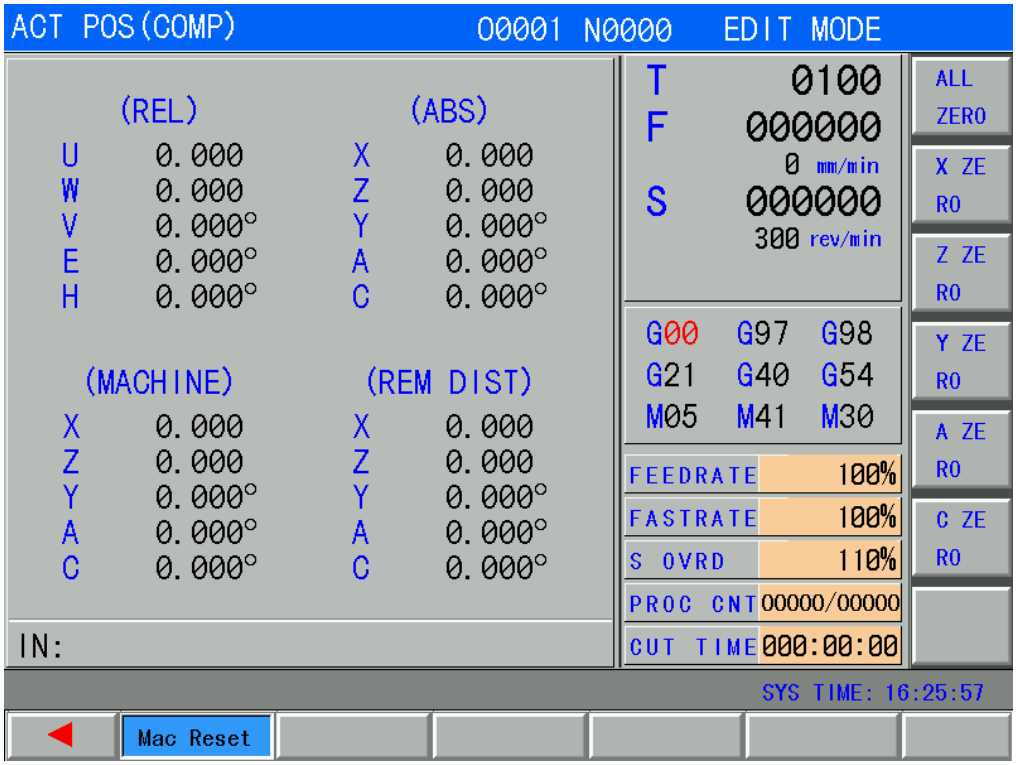
Parameter 5 1.5 [M3 bus spindle speed feedback (0: external encoder 1: M3 bus feedback)];

The parameter 5 1.5 only works when 51.4=1 is valid.

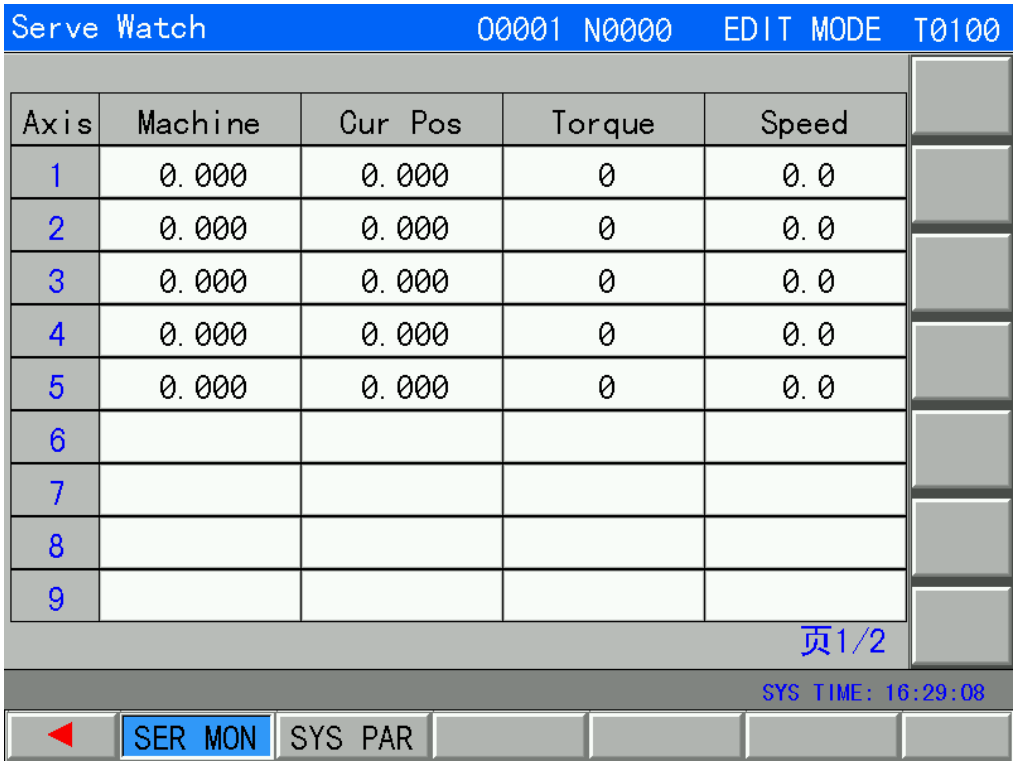
Note: The gear ratio molecule corresponding to the motor drive of 2 3 positions needs to be changed to 64, and after the numerator is changed to 64, the gear ratio setting is consistent with 17 bits; For a direct-connection screw with a pitch of 1 0 mm, the gear ratio of the system parameters is 1/1, and for a direct-connection screw with a pitch of 8mm, the system parameter gear ratio is 5/ 4； The counting method is the same as normal pulse counting.

2) The bus zero setting is the same as the absolute zero setting:

Enter the position interface, find the menu [Machine Clearing] to enter Clear Machine Zero:



3) After setting, you can see if the actual machine coordinates and the coordinates of the motor feedback are opposite



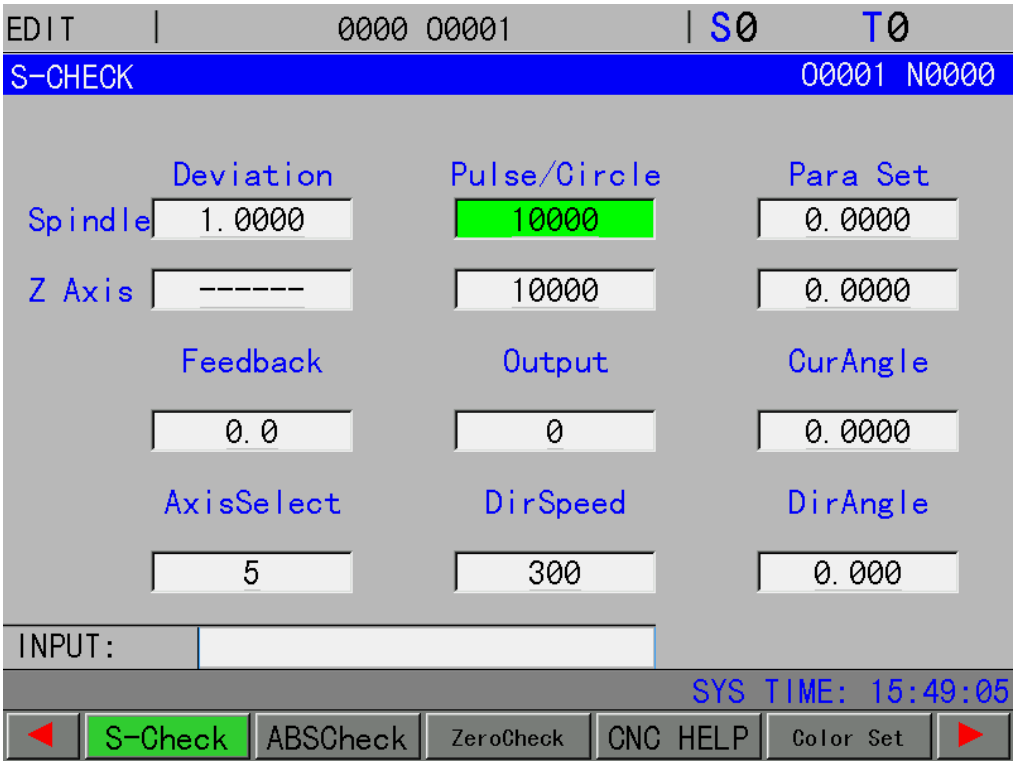
Note: When the coordinates of the machine tool are different from the current coordinates of the motor, it is generally caused by the wrong reading direction of the encoder, and the bit parameter 193 can be modified for

modification.

Appendix 11, Spindle Rigid Tapping Monitoring Feedback

For the use of different spindle servos for rigid tapping, the synchronization error of the spindle and the Z axis will have different deviations, even if the same type of spindle servo different parameter adjustment will appear different deviation values, in order to rigid tapping effect is good, try to minimize the synchronization error between the spindle and the Z axis or use tapping spindle and Z axis compensation.

The deviation value of the spindle from the Z-axis can be viewed in Diagnostics [Spindle detection].



The deviation value of P A9=80 in the Z axis is 1 3.7, so the spindle needs to be adjusted to about 1 3.7 (size range 0.1) The best effect, how to use [number of pulses per revolution] Fill in the number of pulses sent by the system to tap the spindle one turn (generally 10000, number parameters 230=10, 233=1 when 10000), then run the program:

G80G90G54Z20;

M29S100;

G84Z-20R5F100;//(F is programmed with pitch * speed, or direct F = pitch, determined by the number parameter 31.0).

G80;

Switch to the [spindle detection] interface before running, place the cursor at the spindle position, and run the program to view the deviation value; If the deviation value has been unstable and can not be accurately recorded, when the main axis deviation fluctuates greatly, then it is necessary to adjust the spindle servo position gain to reduce its fluctuation value, wait for the deviation value to be recorded accurately, check whether the size of the deviation value is about 1 3.7 (size range 0.1), if there are not two operation modes, 1 Enter the deviation values

of the spindle and Z-axis into their respective [parameter settings]; 2. Adjust the spindle servo position feed-forward parameters and position gain to adjust the spindle deviation value to about 13.7, at this time [parameter setting] does not need to be input (0); The second method is recommended.

Appendix 12, 4/5 Axis follows the Z axis special instructions

Parameters: bit parameter 42.1 G200-G205 follows whether the Z-axis function is valid (0: no, 1: yes).

Command format: G200 The 4th axis moves with the Z axis

G201 Cancel 4th axis follows Z axis

G202 The 5th axis moves with the Z axis

G203 Cancel 5th axis follows Z axis

G204 The 4th and 5th axes follow the Z axis

G205 Cancel 4 and 5 axes follow the Z axis

Note 1: Cancel the follow during alarm, reset, and emergency stop, and at the same time command to follow and cancel the following requires a separate line of instructions;

Note 2: The number of pulses emitted by the following shaft is determined by the Z axis, so when using the follow function, the gear ratio of the following shaft needs to be consistent with the gear ratio of the Z axis;

Usage examples:

G54G90G17G80G40;

G0X0Y0Z20A20; At the same time start following at the absolute position of 20

G200;// Turn on the follow function

G81Z-20R5F120;//The 4th axis follows the Z axis to achieve 4th axis punching

X10Y10;

G80;

G201;//Cancel follow function

Appendix XIII: Automatic screw clamping function M72/M73 during drilling

Parameters: Parameter 1 4.5 G73/G81/ G82/G83/G85/G86/G88/G89 drilling screw automatic clamping (0:no 1:Yes) is valid

Automatic output clamping Y1.5 (M72/M73);

Clamping control parameters T46 M72 output clamping delay (default 200).

T47 M72/M73 code completion delay (default 200).

Working principle: When commanding the drilling instruction [G73/G81/G82/G83/G85/G86/G88/G89], the

XY axis mobile positioning quickly outputs M73 Release the clamp (turn off Y 1.5), wait for M 73 to complete, and then move X Y positioning, after the positioning is completed, output M72 delay output clamp Y 1.5, Wait for the completion of M72, then instruct the Z axis to move and punch holes, and repeat the above actions when there are multiple holes.

Note: After the punching is completed, M73 needs to be instructed to release the lead screw clamping;

Example:

G54G90G17G80G40;

G0X0Y0Z20; //

G73X10Y10Z-20R5F120Q5;//

G80; //Unclamp the lead screw

Note 1: At the end of the G73/G81/G82/G83/G85/G86/G88/G89 commands, output M73 opens the clamp.

Note 2: The M code cannot exist in the end loop code G80 section, otherwise the alarm address will be repeated and error-free.

Note 3: Instructions to the cutting code in the cycle code [such as cutting instructions such as G01/G02/G03, etc.] will cancel the cycle code, so no M code can appear in the first cutting instruction segment. Otherwise, the alarm address is duplicated and incorrect.

NOTE 4: If the cycle instruction [G73/G81/G82/G83/G85/G86/G88/G89] is not used in machining or the cutting instruction [G01/G02/G03] is canceled, run the other instructions [G76/G87/ G74/G84/G22/G23 and other instructions] The lead screw clamping function will not be released.

Appendix XIV, Rotary Axis Control Method (only available in version 2021 or above).

1、 Set the rotation axis parameters:

Parameter 33.0 sets the X-axis type (0: linear axis 1: rotation axis).

Parameter 33.1 sets the Y-axis type (0: linear axis 1: rotation axis).

Parameter 33.2 sets the Z-axis type (0: linear axis 1: rotation axis).

PARAMETER 33.3 SETS THE 4TH AXIS TYPE (0: LINEAR AXIS 1: AXIS OF ROTATION).

PARAMETER 33.4 SETS THE 5TH AXIS TYPE (0: LINEAR AXIS 1: AXIS OF ROTATION).

2、 Axis of rotation type:

The system divides the rotation axis settings into 2 categories, 1) rotation nearby, 2) rotation by symbol;

Parameter 34.2 When the X axis is the axis of rotation (0: rotation in the direction of the symbol 1: rotation nearby).

Parameter 34.3 When the Y-axis is the axis of rotation (0: rotate in the direction of the symbol 1: rotate nearby).

Parameter 35.3 When the Z-axis is the axis of rotation (0: rotate in the direction of the symbol 1: rotate nearby)).

PARAMETER 35.1 WHEN THE 4TH AXIS IS THE AXIS OF ROTATION (0: ROTATE IN THE DIRECTION OF THE SYMBOL 1: ROTATE NEARBY).

PARAMETER 37.1 WHEN THE 5TH AXIS IS THE AXIS OF ROTATION (0: ROTATE IN THE DIRECTION OF THE SYMBOL 1: ROTATE NEARBY).

1) Nearby rotation control:

The nearest rotation control mode machine coordinates and absolute coordinates are cyclic between 0-360 degrees, the relative coordinate coordinates can be set cycle or partial cycle, the shortest path control is performed during absolute programming, and the incremental movement is performed during incremental programming (the shortest distance is not performed).

N1 G90 G0 C0	The sequence number	The amount of actual location	The absolute coordinate value of the end point	Machine coordinates
N2 G0 C300	N2	-60, negative movement	300	300
N3 C100	N3	-160, negative movement	100	100
N4 C360	N4	-100, negative movement	0	0
N5 C-30	N5	-30, negative movement	330	330
N6 C0	N6	30, forward movement	0	0
N7 C360	N7	0, do not move	0	0
N8 C720	N8	0, do not move	0	0
N9 C0	N9	0, do not move	0	0
N10 C360	N10	0, do not move	0	0
N11 C360	N11	0, do not move	0	0
N12 G91 C540	N12	540, forward movement	180	180
N13 G91 C360	N13	360, forward movement	180	180
N14 G91 C-200	N14	-200, negative movement	340	340

2) Select controls by symbol direction:

According to the symbol direction rotation control mode machine coordinates and absolute coordinates are cyclic between 0-360 degrees, the relative coordinate coordinates can be set cycle or partial cycle, according

to the symbol rotation can be divided into two types of rotation axis A / B: (Note: A / B two types are invalid under the nearest rotation control).

Parameter 35.4 sets the X axis to the type of rotation (0: rotation axis A type 1: rotation axis type B).

Parameter 35.5 sets the Y axis to the type of rotation (0: rotation axis A type 1: rotation axis type B).

Parameter 35.6 Set the Z axis to the type of rotation (0: rotation axis A type 1: rotation axis type B).

PARAMETER 34.1 SETS THE 4TH AXIS TO THE TYPE OF ROTATION (0: AXIS OF ROTATION A TYPE 1: AXIS OF ROTATION TYPE B).

PARAMETER 36.1 SETS THE 5TH AXIS TO THE TYPE OF ROTATION (0: AXIS OF ROTATION TYPE A 1: AXIS OF ROTATION TYPE B).

Rotation axis type A: Absolute programming is performed in the symbolic direction, running in a similar way to a linear axis, moving in the positive direction from C 0 to C 300, moving in the negative direction from C 300 to C 200, and performing incremental movement (without shortest distance) during incremental programming.

N1 G90 G0 C0	The sequence number	The amount of actual location	The absolute coordinate value of the end point	Machine coordinates
N2 G0 C300	N2	300, forward movement	300	300
N3 C100	N3	-200, forward movement	100	100
N4 C360	N4	260, forward movement	0	0
N5 C-30	N5	-30, negative movement	330	330
N6 C0	N6	-330, negative movement	0	0
N7 C360	N7	360, forward movement	0	0
N8 C720	N8	720, forward movement	0	0
N9 C0	N9	0, do not move	0	0
N10 C360	N10	360, forward movement	0	0
N11 C360	N11	360, forward movement	0	0
N12 G91 C540	N12	540, forward movement	180	180

N13 G91 C360	N13	360, forward movement	180	180
N14 G91 C-200	N14	-200, negative movement	340	340

Note: In type A/B, when the coordinates go to the multiple of 3 60 or 3 60, the coordinates are calculated from 0 when the next line is immediately followed by the multiple of 3 60 or 3 60; For example, the above program C 0-C360 (at this time walked a lap of 360 degrees) - C720 (at this time from the 0 segment is counted as 2 turns).

Rotation axis type B: Absolute programming is performed in the direction of the symbol, and the operation mode moves in the positive direction (moving in the negative direction when there is a negative symbol), moving in the positive direction from C 0 to C 300, and moving in the positive direction from C 300 to C 200 [300-360(0)-200], incremental movement is performed during incremental programming (minimum distance is not performed).

N1 G90 G0 C0	The sequence number	The amount of actual location	The absolute coordinate value of the end point	Machine coordinates
N2 G0 C300	N2	300, forward movement	300	300
N3 C100	N3	160, forward movement	100	100
N4 C360	N4	260, forward movement	0	0
N5 C-30	N5	-30, negative movement	330	330
N6 C0	N6	60, forward movement	0	0
N7 C360	N7	360, forward movement	0	0
N8 C720	N8	720, forward movement	0	0
N9 C0	N9	0, negative movement	0	0
N10 C360	N10	360, forward movement	0	0
N11 C360	N11	360, forward movement	0	0
N12 G91 C540	N12	540, forward movement	180	180

N13 G91 C360	N13	360, forward movement	180	180
N14 G91 C-200	N14	-200, negative movement	340	340

Note: In type A/B, when the coordinates go to the multiple of 3 60 or 3 60, the coordinates are calculated from 0 when the next line is immediately followed by the multiple of 3 60 or 3 60; For example, the above program C 0-C360 (at this time walked a lap of 360 degrees) - C720 (at this time from the 0 segment is counted as 2 turns).

1. Rotate the negative angle of type A**/B** by symbol

Bit parameter 37.6 Rotate negative angle by sign instruction type 0 (0 0: Type A 01: Type B 10: Type C 11 :D Type).

Bit parameter 37.7 Rotate negative angle by sign instruction type 1 (0 0: Type A 01: Type B 10: Type C 11 :D Type).

1. Negative angle editing does not exceed -360 cases
2. Negative angle editing cases over -360

1) Negative angle A type 37 7=0 、37. 6=0

When not exceeding -360, take the shortest distance in the negative direction;
Above -360, take the negative (linear axis) distance.

N1 G0 C0	The sequence number	The actual amount of movement	Absolute coordinates	Machine coordinates
N2 C-260	N2	-260	100	100
N3 C-260	N3	0	100	100
N4 C-360	N4	-460 (-100-360)	0	0
N5 C-540	N5	-540 (-360-180)	180	180
N6 C-540	N6	-720 (-180-360-180)	180	180
N7 C-260	N7	-80	100	100
N8 C-720	N8	-820 (-100-360-360)	0	0

2) Negative angle **type B 37 7=0 、37. 6=1**

When exceeding or not exceeding -360, take the negative (linear axis) distance.

N1 G0 C0	The sequence number	The actual amount of movement	Absolute coordinates	Machine coordinates
N2 C-260	N2	-260	100	100
N3 C-260	N3	-360 (-100-260)	100	100
N4 C-360	N4	-460 (-100-360)	0	0
N5 C-540	N5	-540 (-360-180)	180	180

N6 C-540	N6	-720 (-180-360-180)	180	180
N7 C-260	N7	-440 (-180-260)	100	100
N8 C-720	N8	-820(-100-360-360)	0	0

3) Negative angle C type 37 7=1 、37. 6=0

When not exceeding -360, take the shortest distance in the negative direction;

When exceeding -360, go in the negative direction for a relatively short distance, do not move with the same command in a row not exceeding -360, and move in a row with the same movement of -360 degrees -360 displacement.

N1 G0 C0	The sequence number	The actual amount of movement	Absolute coordinates	Machine coordinates
N2 C-260	N2	-260	100	100
N3 C-260	N3	0	100	100
N4 C-360	N4	-100	0	0
N5 C-540	N5	-180	180	180
N6 C-540	N6	-360 (-180-180)	180	180
N7 C-260	N7	-80	100	100
N8 C-720	N8	-100	0	0

4) Negative angle D type 37 7=1 、37. 6=1

When exceeding or not exceeding -360, take the shortest distance in the negative direction;

N1 G0 C0	The sequence number	The actual amount of movement	Absolute coordinates	Machine coordinates
N2 C-260	N2	-260	100	100
N3 C-260	N3	0	100	100
N4 C-360	N4	-100	0	0
N5 C-540	N5	-180	180	180
N6 C-540	N6	0	180	180
N7 C-260	N7	-80	100	100
N8 C-720	N8	-100	0	0

Appendix XV, In-Program Call Subroutine Function (available in subsequent versions 2021.12).

Note: On the basis of the original M98P_ call subroutine, add the usage of calling subroutine within the program (the main program and the subprogram are in the same program).

Parameters: bit parameter 29.6 M98 H_L_ subroutine function (0: invalid 1: valid) [invalid by default]. ；

Format: M98 H_ L_;

Description: H_ subroutine start line number;
L_ Number of calls;

When L_ is omitted, it is invoked once by default
The subroutine ends M99, and the subroutine returns to execute the next line of the main program;

Note 1: The subroutine must end with M99, otherwise it will be called incorrectly;
Note 2: When the subroutine does not end with M99, the subroutine is only called once, and the whole program ends at the same time;
Note 3: Subroutines are nested up to 10 levels;

Example:

```
O0001;
M3S1000;
....
....
M98 H10 L5;// call the subroutine after N10 5 times
N5 ....
   ....
M98 H20 L5;// call the subroutine after N20 5 times
N6 ....
N7 ....
......

M30;// End of procedure
N10;// Subroutine D10
....
....
M99;// returns line N5 after the end of the subroutine
N20;// Subroutine D20
....
....
M99;// The end of the subroutine returns line N6
```

Appendix XVI, New Modifications to Handwheel Test Cut Simulation Speed Control (available in subsequent versions 2021.12).

When the new handwheel simulation is applied, the feed shaft speed is modulated according to the frequency of handwheel shaking, but the upper speed limit is still affected by number parameter 149 and number parameter 82;

Parameters: bit parameter 1 8.7 [handwheel test cut follows handwheel rotation speed (0: invalid 1: valid)];
Number parameter 1No. 53 [Handwheel test cut follows the speed change base];

The larger the base of the handwheel test cut following the speed change, the faster the speed change (equivalent to the greater the acceleration and deceleration, the more obvious the jitter); The smaller the base of change, the slower the change (equivalent to smaller acceleration and deceleration, smaller jitter).

Appendix XVII, Bus may have 6/7 axis parameters for the 6/7 axis hardware version (2021.12 subsequent versions have).

Note: Standard ladder diagram to write the 6th axis manual/handwheel common button, through the function button to switch the 5th axis or 6th axis;
Open the 6th axis function by number parameter 303.

Note: The 7th axis is used as an auxiliary axis, only the pulse spindle and **tapping shaft functions, and the pulse spindle only has the first spindle function G25.4/G25.5 function;**
No coordinates are displayed.

Note: Spindle port assignment with Axis 5 or Axis 6 or Axis 7 hardware;

1: The normal version hardware has 5 axes, and the spindle port can choose the 4th axis or the 5th axis;
2: The TI hardware version has 6 axes, the spindle port can be selected as a 4th axis or 5th axis or 6th axis;
3: The bus version hardware has 6 feed axes and 1 auxiliary axis, and the auxiliary axis is the 7th axis, which only has the function of pulse spindle and tapping;

The port selection has parameters to determine:
Number parameter 326 Spindle pulse port selection (4:4th 5 or other: 5th 6:6th 7:7th) (default 5th axis).
Bit parameter 5 0 Rigid tapping selects axis (0: 5th axis 1: 4th axis) (default 5th axis).
Bit parameter 5 2 Rigid tapping selects the 6th axis (0: invalid 1: effective).
Bit parameter 5 7 Rigid tapping selects the 7th axis (0: invalid 1: valid).

Other selections have no effect when selecting axis 7, and axis 4/5 are invalid when axis 6 is selected

1. Tap to select the axis

Parameter 5.0 [tapping selection (0: 5th axis 1: 4th axis)].
Parameter 5.2 [tapping selection 6th axis (0: invalid 1: valid)].
Parameter 5.7 [tapping selection 7th axis (0: invalid 1: valid)].

When selecting the 6th axis is valid, selecting the 4/5 axis is invalid;
When selecting the 7th axis is valid, selecting the 4/5/6 axis is invalid;

Tapping teeth ratio is 2 33-233/233-236
Effective tapping monitoring;

2. Feed shaft and analog spindle settings

Parameter 7.0 Axis 6 sets the analog spindle (0: invalid 1: valid).
Note: PLC signal G25.4/G25.5 forward and reverse

Parameter 7.1 6th axis analog spindle (0: 1st spindle 1: 2nd spindle).
Note: PLC signal G26.2/G26.3 2nd spindle signal/G35.6 analog voltage signal

Parameter 8.5 6th axis movement direction (0: not reverse 1: negation) can control the forward and reverse rotation of the spindle and the tapping direction
Parameter 20.5 Whether the 6th axis move key is negated (0:No 1:Yes)
Number parameter 95 6th axis fast moving speed
Number parameter 356 6th axis unidirectional positioning direction and overtravel

Number parameter 420 6th axis molecule
Number parameter 421 6th axis denominator
NUMBER PARAMETER 442 6TH AXIS FRONT ACCELERATION AND DECELERATION L-TYPE TIME CONSTANT (ANALOG SPINDLE
NUMBER PARAMETER 443 6TH AXIS FRONT ACCELERATION AND DECELERATION S-TYPE TIME CONSTANT
NUMBER PARAMETER 444 6TH AXIS REAR ACCELERATION AND DECELERATION L-TYPE TIME CONSTANT
NUMBER PARAMETER 445 6TH AXIS REAR ACCELERATION AND DECELERATION E-TYPE TIME CONSTANT

Number parameter 446 6th axis manual fast movement speed
Number parameter 447 6th axis cutting feed maximum control speed

Number parameter 439 The amount of backlash compensation for the 6th axis
Number parameter 440 The compensation step size of the 6th axis gap compensated in a fixed frequency manner
Number parameter 441 6th axis reverse positioning tolerance

Number parameter 451 Reserve the pitch error compensation number for the reference point of the 6th axis

Number parameter 452 Reserved 6th axis pitch error compensation points
Number parameter 453 reserved 6th axis pitch error compensation multiplier

Note: [6th axis G0 / manual use of acceleration and deceleration type to follow other axes]

Parameter 3 3.6 The 6th axis is set to (0 linear axis 1: axis of rotation).
Parameter 36.2 When the axis of rotation is set to the axis of rotation for the 6th axis, it is (0: rotate in the direction of the symbol 1: rotate nearby).
Parameter 3 6.3 The 6th axis is the axis of rotation (0: rotation A type 1: rotation type B).
Bit Parameter 36.4 Whether the 6th axis requires a coordinate system tool interface (0: no 1: required).

3. Absolute value setting

Bit parameter 4.5 Whether the 6th axis reads absolute values (0:yes 1:no).
The 6th axis reads the absolute value coordinates (0: do not negate 1: negate).
Parameter 39.5 The 6th axis reads the absolute encoder number (0:17 bits 1:23 bits).
NUMBER PARAMETER 422 6TH-AXIS SERVO COMMAND MULTIPLIER FACTOR (CMR).
NUMBER PARAMETER 423 6TH AXIS SERVO COMMAND DIVISION FACTOR (CMD).

4. Zero and limit settings

Parameter 26.5 The 6th axis returns to the machine in the direction of zero (0: positive direction 1: opposite direction).
Parameter 28.7 6th axis G28 back reference point (0:no 1:yes) every time the stop is used
Parameter 29.0 When the 6th axis returns to zero (0:No 1:Yes), a one-turn signal is used to slow down to zero
Number parameter 388 6 axis zero off the stop to detect a turn signal distance
Number parameter 448 The 6th axis returns to the zero high-speed speed of the machine
Number parameter 450 The grid offset or reference point offset for axis 6

Parameter 29.1 6th axis shield soft limit alarm (0: invalid 1: valid).
Parameter 2 9.2 6th axis shield hard limit alarm (0: invalid 1: valid).

Number parameter 435 Maximum travel in the negative direction of the 6th axis
Number parameter 436 Maximum travel in the positive direction of the 6th axis
Number parameter 437 Maximum stroke in the negative direction of the 6th axis (second stroke limit).
Number parameter 438 The maximum stroke in the positive direction of the 6th axis (second stroke limit).

NUMBER PARAMETER 431 6TH IS THE FIRST REFERENCE VALUE ON THE MECHANICAL COORDINATE SYSTEM
NUMBER PARAMETER 432 6TH IS THE 2ND REFERENCE VALUE ON THE MECHANICAL COORDINATE SYSTEM
NUMBER PARAMETER 433 6TH 3RD REFERENCE VALUE ON THE MECHANICAL COORDINATE SYSTEM
NUMBER PARAMETER 434 6TH REFERENCE VALUE ON THE MECHANICAL COORDINATE SYSTEM

5. Coordinate system records

NUMBER PARAMETER 424 EXTERNAL WORKPIECE ORIGIN 6TH AXIS OFFSET
Number parameter 425 The workpiece origin offset of the workpiece coordinate system 1 (G54_6TH).

Number parameter 426 The workpiece origin offset of the workpiece coordinate system 2 (G55_6TH).
Number parameter 427 The workpiece origin offset of the workpiece coordinate system 3 (G56_6TH).
Number parameter 428 The workpiece origin offset of the workpiece coordinate system 4 (G57_6TH).
Number parameter 429 The workpiece origin offset of the workpiece coordinate system 5 (G58_6TH).
Number parameter 430 The workpiece origin offset of the workpiece coordinate system 6 (G59_6TH).

7th axis parameters:

Parameter 2.0 The 7th axis sets the analog spindle (0: invalid 1: valid) signal G 25.4/G25.5 forward and reverse

NUMBER PARAMETER 96 7TH-AXIS PULSE SPINDLE MOLECULE

NUMBER PARAMETER 97 7TH AXIS PULSE SPINDLE DENOMINATOR

NUMBER PARAMETER 98 7UTH FAST FORWARD ACCELERATION AND DECELERATION LINEAR TYPE/PULSE SPINDLE ACCELERATION AND DECELERATION TIME CONSTANT

Appendix XVIII, System Control Spindle Positioning Method (2 021.12 subsequent versions have).

Note: Use ordinary feed servo for spindle positioning or use the system position mode to generate pulse spindle positioning function, or bus servo **spindle function to use spindle positioning.**

Function description: feed servo as spindle or spindle servo is to use servo internal positioning function or M3 bus servo spindle positioning, the system by sending out pulse form to make the spindle position mode to move to the positioning angle, no need to connect the positioning completion signal.

Relevant parameters:

Parameter 36.6 [Whether the spindle impulses in direction (0:no 1:Yes)] The direction of direction specifies the positive direction. 〉

Bit parameter 38.6 [spindle encoding feedback direction (0:forward 1:reverse)].

Note: When the feedback direction of the spindle encoder is opposite to the direction of spindle movement, the spindle is positioned to alarm;

NUMBER PARAMETER 297 [SPINDLE ORIENTATION AXIS SETTING (DEFAULT 0)] 4 FOR 4TH AXIS, 5 FOR OTHERS 5 TH AXIS, 6 FOR 6TH AXIS

Number parameter 298 [spindle orientation speed (default 100)].

Number parameter 299 [spindle orientation angle (default 0)].

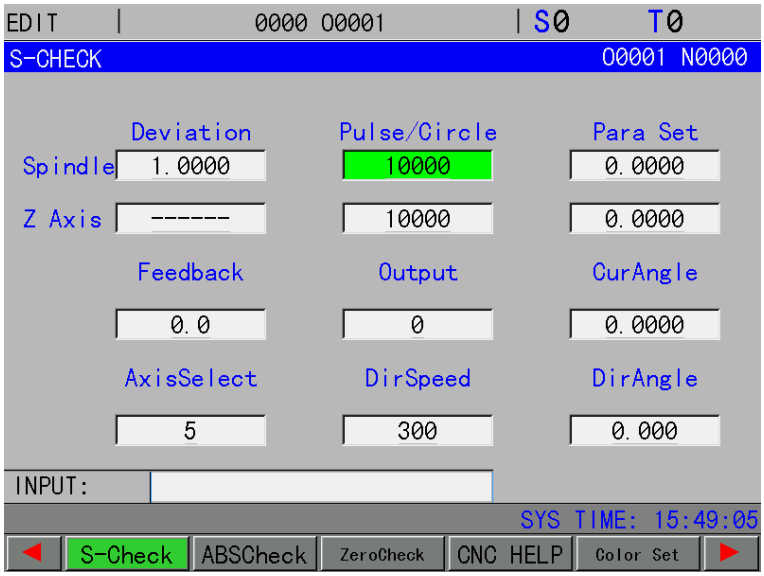
The above parameters require level 2 password modification

After turning on this function, you can see the corresponding parameters in the spindle detection of the diagnostic interface, and the current spindle position, directional speed, directional angle, directional axis and

other information can be displayed.

The spindle does not need to be set to the CS axis;

The number of positioning axes does not need to be turned on:



Note 1: When using this function, there must be spindle encoder feedback, and the number of spindle encoder lines [number parameter 201] and encoder to spindle ratio [number parameter] need to be set 220/number parameter 223];

NOTE 1: Spindle and encoder 1:1 or 2:1 [spindle turns 2 turns encoder 1 turn]. In the case of just using the encoder Z phase zero point (set the directional axis to use a rotation signal as a deceleration signal); When the ratio is 1:2, the spindle turns 1 turn The encoder turns 2 turns, and for those that only need to be oriented angles, in spindles 0 and 1 80 The degree is the same straight line, which has no effect on the directional tool change;

For example, when using the 5th axis as the directional axis of the main axis [number parameter 297=5], the scale is suitable for the use of a revolution signal, and the 5th is set When the shaft returns to zero, the rotation signal is used as the deceleration signal [parameter 28.4=1];

Parameter 028.0 When the X axis returns to zero (0:No 1:Yes), a turn signal is used to slow down to zero

Parameter 028.1 When the Y-axis returns to zero (0:No 1:Yes), a rotation signal is used to slow down to zero

Parameter 028.2 When the Z-axis returns to zero (0:No 1:Yes), a revolution signal is used to slow down to zero

Parameter 028.3 When the 4-axis is reset to zero (0:No 1:Yes), a one-turn signal is used to slow down to zero

Parameter 028.4 When the 5-axis is returned to zero (0:no 1:yes), a one-turn signal is used to slow down to zero

Parameter 29.0 When the 6th axis returns to zero (0:No 1:Yes), a one-turn signal is used to slow down to zero

Note 2: The spindle and the encoder are other ratios, it is necessary to use the switching signal and the

encoder Z signal to determine the zero point position, the system first checks the corresponding shaft deceleration signal falling edge and then detects the Z phase zero point (setting the directional axis can not use a rotation signal as a deceleration signal).

For example, when the 5th axis is used as the directional axis of the spindle [number parameter 297=5], the ratio is suitable for the use of switching signals and a turn signal, and the 5th is set When the shaft returns to zero, a rotation signal is not used as a deceleration signal [bit parameter 28.4=0], and the spindle positioning sensor is connected to the 5th shaft deceleration signal;

The K22.4 X-axis deceleration signal is valid with +24V (0:disconnect 1:on);
The K22.5 Z-axis deceleration signal is valid with +24V (0:disconnect 1:on);
The K22.6 Y-axis deceleration signal is valid with +24V (0:Disconnect 1:On);
The K22.7 4th axis deceleration signal is valid for +24V (0:Disconnect 1:On).
K22. 3 The 5th axis deceleration signal is valid with +24V (0: off 1: on).
K22. 2 The 6th axis deceleration signal is valid with +24V (0: off 1: on).

When setting the positioning angle, first adjust the required spindle positioning speed [number parameter 298] (different positioning speeds, positioning angles are different, so the positioning speed can generally not be changed after setting, when the positioning speed needs to be changed, the positioning angle needs to be readjusted); The spindle positioning angle is set to 0 [number parameter 2 99], the spindle first performs a complete positioning, then releases the positioning, manually rotates the spindle to the required angle, views the spindle feedback angle, and records the spindle positioning angle parameter 2 99 。

Note: When you need to adjust the new positioning angle, you need to set the spindle positioning angle to 0 [number parameter 2 99], the spindle first performs a complete positioning, then releases the positioning, manually rotates the spindle to the required angle, views the spindle feedback angle, and records in the spindle positioning angle parameter 299 。

When the system only performs M19 or manual orientation, the system orientation angle is parameter 299 angle.
When doing M19 S_; //S is the angle value 0-360 [is relative to the starting positioning angle] **[the function is temporarily unavailable].**
The system orientation angle is S_; **[Feature temporarily unavailable].**

When the spindle is in the free state, the system spindle coordinates change with the spindle encoder.
When the servo spindle is also applicable to this function, it is necessary to connect the spindle orientation signal to the driven position mode, and there is no need to connect the spindle orientation completion signal.

Note: M3 spindle drive setting, bit parameter 51.4 [M3 communication last address as bus spindle function (0: invalid 1: valid)];
Parameter 5 1.5 [M3 bus spindle speed feedback (0: external encoder 1: M3 bus feedback)].

Appendix XIX: Summary of System Parameters (2023 Edition).

Bit parameters:

No. 1

001. 0 (0:metric 1:imperial) input

001.3 (0: single step 1: handwheel) mode

001.4 Spindle speed (0: analog voltage control 1: switching control).

No. 2

002.0 Whether axis 7 is set to pulse spindle (0:No 1:Yes).

002.5 Pulse mode selection (0: pulse + direction 1: quadrature pulse).

002.6 pulse width (0: auto width 1:2.5 μs).

002.7 Spindle forward and reverse (0: signal control 1: voltage control).

No. 3

003. 0 Metric conversion is automatic when the knife complement value (0:no 1:) is converted

003.1 The workpiece coordinate system value (0:no 1:) is automatically converted when the metric system is converted

003.2 Manual clearing of mechanical coordinates (0: invalid 1: valid).

003.3 Enter letters + numbers when the coordinate system is set against the tool (0: offset zero point 1: absolute coordinates).

003.4 Select the type of tool length compensation (0: method A 1: mode B).

003.5 Pitch error compensation function (0: invalid 1: valid).

003.6 Tool row function selection (0: invalid 1: valid).

No. 4

004. 0 The X-axis (0:Yes 1:No) reads absolute values

The 004.1 Y-axis (0:Yes 1:No) reads absolute values

004.2 The Z-axis (0:Yes 1:No) reads absolute values

004.3 The 4th axis (0:Yes 1:No) reads absolute values

004.4 The 5th axis (0:Yes 1:No) reads absolute values

004.5 Axis 6 (0:Yes 1:No) reads absolute values

004.6 Minimum moving unit 0: 0.001, 0.0001; 1:0.0001,0.00001(mm/deg,inc)

004.7 (0:metric 1:imperial) output

No. 5

005. 0 Rigid tapping selection axis (0: 5th axis 1: 4th axis).

005.1 When the knife makes up C or there is no intersection point in the chamfer (0:Yes1:No), the calculation continues as the shortened type

005.2 Rigid tapping selects axis 6 (0: invalid 1: valid).

005.3 M02 begins with the cursor (0:Do not return 1:Return).

005.4 M30 cursor (0:not return, 1:return) goes back to the beginning

005.5 The program cursor (0:No 1:Yes) is fixed in the middle

005.6 Program editing (0:Yes 1:No) supports block deletion

No. 6

006. 0 back to zero mode (0: after the block 1: before the gear).

006.1 Location interface single-segment entry procedure (0:valid 1:invalid)

006.3 Zero return mode (0:none 1:Yes) one turn signal

006.4 Zero return mode selection when there is no turn signal (0:A mode 1:B mode).

006.5 Drive communication mode (0: incremental 1: absolute value 485 communication).

006.6 Absolute values are updated after the program finishes (0:No 1:Yes).

006.7 The boot position page is displayed (0:Chengjian 1:Comprehensive).

No. 7

007. 0 Whether the sixth axis is set to the pulse spindle (0:No 1:Yes).

007.1 The sixth axis is set as a pulse spindle (0:1st spindle 1:2nd spindle).

007.2 Draw a full circle (0:Yes 1:No) Special treatment

007.3 When the feed shaft stops (0:no 1:Yes) when the pulse spindle stops, it stops in the pre-start position

007.4 Whether the 5th axis is set to the pulse spindle (0:No 1:Yes).

007.5 Whether the 4th axis is set to the pulse spindle (0:no 1:Yes).

007.6 Diagnostic interface display toggle

007.7 The boot screen (0:No 1:Yes) is displayed in full screen

No. 8

008. 0 X axis moves the coordinate direction (0: do not negate 1: negation).

008.1 Y axis moving coordinate direction (0: do not negate 1: negation).

008.2 Z axis moving coordinate direction (0: do not negate 1: negation).

008.3 4th axis shift coordinate direction (0: do not negate 1: negation).

008.4 5th axis shift coordinate direction (0: do not negate 1: negation).

008.5 6th axis shift coordinate direction (0: do not negate 1: negation).

008.6 7th axis moves the coordinate direction (0: do not negate 1: negation).

No. 9

009. The 0 X-axis driver alarm signal is a (0:high 1:low) level alarm

The 009.1 Y-axis driver alarm signal is (0:high 1:low) level alarm

The 009.2 Z-axis driver alarm signal is (0:high 1:low) level alarm

009.3 The 4th axis driver alarm signal is (0:high 1:low) level alarm

The 009.4 5th axis driver alarm signal is (0:high 1:low) level alarm

009.7 The spindle driver alarm signal is (0:high 1:low) level alarm

No. 10

010. 0 Whether the parameter setting switch is on when the system is turned on (0:Yes1:No).

010.1 The program code color setting (0:No 1:Yes) is valid

010.2 Defined

010.3 Whether the PLC parameter backup and restore function is supported (0:No 1:Yes)?

010.4 Arc programming maximum speed limit (0: invalid 1: valid).

010.5 Machine number randomly generated (0: valid 1: invalid).

010.6 Manually switch to another way (0:not 1:Required) to wait

Number 11

011. 0 The backlash is incorrect (0:No 1:Yes) executes the next segment

011.1 Whether the program set switch on when booting (0:Yes1:No).

011.3 High-speed machining forward processing (0:no 1:Yes) open

011.4 Turn on the angular speed optimization function during machining (0:no 1:Yes).

011.5 Emergency stop (1:no 0:required) returns to zero

011.6 System power-on (0:Yes 1:No) sets the spindle speed

011.7 Anti-clearance compensation method (0: fixed frequency 1: acceleration and deceleration).

Number 12

012. 0 Fast movement before zero (0:neither1:Yes) works

012.1 Read-ahead mode cutting feed acceleration and deceleration smoothing (0: invalid 1: valid).

012.2 High-speed machining (0:No 1:Yes) open

012.3 Installment password level (0: machine tool factory 1: system manufacturer).

012.4 JOG operation acceleration and deceleration (0: linear type 1: stepped type).

012.5 Read-ahead mode with step speed smoothing (0: invalid 1: valid).

012.6 Enter the installment password as (0: with registration code 1: without registration code).

Number 13

013. 0 The coordinates increase when the handwheel (0:reverse 1:forward) turns clockwise

013.1 Number of M99 pieces performed (0:No 1:Yes) plus 1

013.2 Press the number of reset keys to complete the processing (0:not cleared 1:zero)

013.3 Read file format type (0:generic 1:Finely carved).

The 013.5 axis enable signal (0:No 1:Yes) is controlled by a PLC

013.6 The handwheel test cut function is valid for the current segment reverse (0:no 1:yes).

013.7 Whether the rotation displacement of the handwheel disc is fully operational (0:No 1:Yes)

No. 14

014. 0 Positioning (G00) The interpolated trajectory is (0: non-linear 1: linear).

014.1 When fast feeding, when the fast feed multiplier is Fo (0: no stop 1: stop).

014.2 Check for accurate positioning of the row knife (0:No 1:Yes).

014.3 Fallback function after rigid tapping stops (0: valid 1: invalid).

014.4 G73-G89 commands manual intervention to move after other axes are interpolated (0:Yes 1:No).

014.5 G73/83/81/82/85/86/88/89 Automatic clamping of drilling screws (0:no 1:Yes) is valid

014.6 Whether the maximum control speed of the cutting feed is independent for each axis (0:no 1:Yes).

014.7 Cutting feed rate (0: read program F value 1: given internally).

No. 15

015.0 Mechanical zero point (0: no memory 1: memory).

015.1 G28 command when reference point is not established (0: use block 1: alarm).

015.2 Zero not returned except G28 (0: no alarm 1: alarm).

015.3 After the reference point is memorized, the reference point is manually returned to the reference point as (0:G0 fast 1:zero high-speed) speed

015.4 The return to workpiece origin function (0:Yes 1:No) requires the machine to return to zero

015.7 Manual zero return (0:Yes 1:No) selects multiple axes simultaneously

Number 16

016. 0 Fast running mode (0: front acceleration and deceleration 1: rear acceleration and deceleration).

016.1 Acceleration and deceleration before fast running (0: linear type 1: S type).

016.2 Acceleration and deceleration after fast running (0: linear 1: exponential).

016.3 Cutting feed mode in non-read-ahead mode (0: front acceleration and deceleration 1: rear acceleration and deceleration).

016.4 Acceleration and deceleration before cutting feed in non-read-ahead mode (0: linear type 1: S type).

016.5 Acceleration and deceleration after cutting feed in non-read-ahead mode (0: linear 1: exponential).

016.6 JOG operation selection (0:straight 1:exponential) type acceleration and deceleration

016.7 Handwheel running selection (0:straight 1:exponential) type acceleration and deceleration

Number 17

017. 0 Select the processing method (0: non-read-ahead 1: read-ahead).

017.1 Acceleration and deceleration mode after interpolation in read-ahead mode (0: linear type 1: exponential type).

017.2 Pre-read automatic corner deceleration function (0: angle control 1: speed difference control).

017.3 Whether the read-ahead method is detected in place (0:no 1:Yes).

017.4 Whether the acceleration and deceleration program segment overlap imputation before the read-ahead mode is valid (0:no 1:Yes)

017.7 Whether the G1 line drops to corner velocity (0:no 1:Yes).

Number 18

018.0 Macro program public variable #100~#199, after reset (0: not cleared 1: cleared).

018.1 Macro program local variables #1~#50, after reset (0: do not clear 1: clear)

018.3 Whether the single piece machining time cutting is automatically cleared to zero (0:no 1:Yes)

018.4 Machining cutting time power off (0:no 1:yes) cleared

018.6 The judgment range of arc connection arc knife in the same direction is expanded by 0.00005 (0: valid 1: invalid).

018.7 Handwheel test cut follows handwheel rotation speed (0: invalid 1: valid) [change base 153].

Number 19

019.1 G43/G44/G49 length compensation requires moving axes (0:Yes 1:No).

019.2 Whether to perform radius compensation interference check (0:no 1:Yes)

019.3 Starting and retracting forms in tool radius compensation (0:A type, 1:B type).

019.4 G28, G30 command to move to the middle point, (0: do not cancel 1: cancel) radius compensation

019.5 G28, G30 moves to the midpoint, cancels knife time (0: standard action 1: vertical).

019.6 Tool radius compensation amount (0: radius value 1: diameter value) setting

019.7 In radius compensation, whether the corner arc function is valid (0:no 1:Yes)

Number 20

020. 0 X axis movement direction button (0: negate 1: do not negate).

020.1 Y axis movement direction button (0: negation 1: no negation).

020.2 Z-axis movement direction button (0: negate 1: do not negate 1):

020.3 4th axis movement direction button (0: negate 1: do not negate).

020.4 5th axis movement direction button (0: negation 1: no negation).

020.5 6th axis movement direction button (0: negation 1: no negation).

020.6 Spindle SAR signal before cutting (0: do not check 1: check).

020.7 When cutting feed, (0:Allow 1:Not allowed) the spindle stops rotating

Number 21

021. 0 External loop start (0: valid 1: invalid).

021.1 External pause signal (0: valid 1: invalid).

021.2 Emergency stop alarm signal (0: check 1: do not check).

Number 22

022.0 Exclusion zone at the second stroke limit (0:Inside 1:Outside)

022.1 Soft limit function before zero return (0: invalid 1: valid).

022.2 When issuing the overtravel command, alarm in overtravel (0:front 5.0mm 1:after).

022.3 Machine tool zero point mode (0: check 1: do not check) soft limit alarm

022.4 The 4th axis is the analog spindle selection (0: 1st spindle 1: 2nd spindle).

022.5 The 5th axis is the analog spindle selection (0: 1st spindle 1: 2nd spindle).

022.6 G17 plane, G02/G03 command K time (0: no alarm 1: alarm).

022.7 In the same paragraph, when two or more identical addresses are instructed (0: no alarm 1: alarm).

Number 23

023. 1 Tool instrument function (0: valid 1: invalid) [cannot be set].

023.2 Tool setting instrument function (0:1 knife 1:Multiple knives)

023.5 Z-axis tool change slow interval speed limit (0: invalid 1: valid).

023.6 X-axis tool change slow interval speed limit (0: invalid 1: valid).

023.7 Y-axis tool change slow interval speed limit (0: invalid 1: valid).

Number 24

024. 1 G31 signal alarm is not received when executing G31 (1: valid 0: invalid).

024.2 Clear the base offset value when manually re-aligning the coordinate system (0:No 1:Yes).

024.3 G08 Modify the radius compensation value type (0: absolute assignment 1: incremental assignment).

024.4 Machine one-key zero function (0: invalid 1: valid).

024.5 The zeroing sequence when the machine tool is valid when one key to zero is valid (0:Z-XY45 1:Z4-XY5)

024.6 The zero-return grid offset speed moves at (0: zero low speed 1: each axis returns to zero high speed).

Number 25

025. 0 position & program monitoring display (0: relative coordinates 1: remaining movement).

025.1 Press the edit key (0:No 1:Yes) to enter the program interface

025.2 Alarm (0:No 1:Yes) Cut to the alarm interface

025.5 Switch to automatic mode (0:no 1:yes) to return to the beginning of the program

025.6 Pressing the reset key to return to the beginning of the program is valid in (0:edit 1:other) mode

025.7 Function ATAN, ASIN range (0:-90.0~90.0 1:90.0~270.0).

No. 26

026. The 0 X axis returns to the machine in the direction of the zero point (0: positive direction 1: opposite direction).

The 026.1 Y-axis returns to the machine in the direction of the zero point (0: positive direction 1: opposite direction).

026.2 The Z-axis returns to the machine in the direction of the zero point (0: positive direction 1: opposite direction).

026.3 The 4th axis returns to the machine in the zero direction (0:positive direction 1:opposite direction).

026.4 The 5th axis returns to the machine in the zero direction (0:positive direction 1:opposite direction).

026.5 The 6th axis returns to the machine in the zero direction (0: positive direction 1: opposite direction).

026.6 Zero detection function (0:No 1:Yes) is valid

026.7 Zero detection function (0:detection 1: self-learning).

Number 27

027. 0 X-axis shielded soft limit alarm (0: invalid 1: valid).

027.1 Y-axis shielded soft limit alarm (0: invalid 1: valid).

027.2 Z-axis shielding soft limit alarm (0: invalid 1: valid).

027.3 4-axis shielded soft limit alarm (0: invalid 1: valid).

027.4 5-axis shielded soft limit alarm (0:invalid 1:valid)

027.5 X-axis G28 back to the reference point (0:no 1:Yes) every time the stop is used

027.6 Y-axis G28 back to the reference point (0:no 1:Yes) every time the stop is used

027.7 Z-axis G28 back to the reference point (0:no 1:Yes) every time the stop is used

Number 28

028. When the 0 X-axis returns to zero (0:no 1:Yes), a one-turn signal is used to slow down back to zero

028.1 When the Y-axis returns to zero (0:No 1:Yes), a one-turn signal is used to slow down to zero

028.2 When the Z-axis returns to zero (0:no 1:yes), a one-turn signal is used to slow down to zero

028.3 When the 4-axis returns to zero (0:no 1:yes), a one-turn signal is used to slow down to zero

028.4 When the 5-axis returns to zero (0:no 1:Yes), a one-turn signal is used to slow down to zero

028.5 4TH axis G28 back reference point (0:no 1:yes) every time the stop is used

028.6 5TH axis G28 back to the reference point (0:no 1:yes) every time the stop is used

028.7 6TH axis G28 back reference point (0:no 1:yes) every time the stop is used

Number 29

029. 0 When the 6-axis is reset to zero (0:No 1:Yes), a one-turn signal is used to slow down to zero

029.1 6-axis shielded soft limit alarm (0: invalid 1: valid).

029.2 6-axis shielded hard limit alarm (0: invalid 1: valid).

The 029.3 Speed page (0:No 1:Yes) is displayed

029.4 Whether to prohibit program editing with program number 8000~8999 (0:No 1:Yes)

029.5 Whether to prohibit program editing with program numbers 9000~9999 (0:No 1:Yes)?

029.6 M98 H_L_ calling in-program subroutine (0:No 1:Yes) is valid

029.7 Launcher (0:No 1:Yes) clears the generated drawing

No. 30

030. 0 PLC axis control function (0: invalid 1: valid).

030.1 PLC axis fast moving speed take (0: parameter setting value 1: input value).

030.2 The 4th axis is set to the PMC axis (0: valid 1: invalid).

030.3 3rd reference point return type

030.5 Reference Point 4 return type

030.6 CAD drawing reading origin (0: CAD origin 1: drawing cross point).

030.7 Returns reference point in automatic mode (0: invalid 1: valid).

Number 31

031. 0 Rigid tapping pitch F setting (0:No 1:Yes).

031.1 CAD (DXF file) punching and tapping function (0:No 1:Yes) is valid

031.2 Add M00 instruction under the CAD Generator Tool Change Instruction M06T_ (0:Yes 1:No).

031.3 System screen H screenshot function (0: invalid 1: valid).

031.4 CAD default tool number (0:1 knife 1:100 cutter).

031.5 When sorting points, the less moving axis starts from the coordinates of the numeric value (0:small 1:large).

031.6 When sorting points, the multi-moving axis starts from the coordinates of the numeric value (0:small 1:large).

031.7 The hyperactive axis is the (0:Y 1:X) axis when sorting dots

No. 32

032. 0 Chinese file displays the actual program name (0: invalid1: valid).

032.1 No M30% no alarm at the end of the program (0: invalid 1: valid).

032.2 983MC keyboard type (0: no digital tube display 1: with digital tube display).

032.3 Reading macro program alarms/macro variable comments (0:No 1:Yes) is valid

032.4 When G20/G21 metric switching cannot be used (0: alarm 1: no alarm).

032.5 Update absolute value success prompt (0: not displayed 1: Display).

032.6 Servo monitoring interface (0: not displayed 1: Display).

032.7 CAD (DXF file) material selection function (0: invalid 1: valid).

No. 33

The 033.0 X-axis is set to (0: Line axis 1: Axis of rotation).

The 033.1 Y axis is set to (0: Line axis 1: Axis of rotation).

The 033.2 Z-axis is set to (0: linear axis 1: axis of rotation).

033.3 Set the 4th axis to (0: linear axis 1: rotation axis).

033.4 Set the 5th axis to (0: linear axis 1: rotation axis).

033.5 Indexing instruction (0:G90/G91 specifies 1:Absolute instruction).

033.6 Set the 6th axis to (0: linear axis 1: rotation axis).

No. 34

034. 1 The type when setting the 4th axis as the axis of rotation (0: axis of rotation A type 1: axis of rotation type B).

034.2 When the X axis is the axis of rotation (0: rotation in the direction of the symbol 1: rotation nearby).

034.3 When the Y axis is the axis of rotation (0: rotate in the direction of the symbol 1: rotate nearby).

034.4 Cs axis function of the Y-axis (0: invalid 1: valid).

034.5 Cs axis function for 4th axis (0: invalid 1: valid).

034.7 Whether it is a four-axis linkage system (0:no, 1:yes)

No. 35

035.0 Absolute coordinate loop function (0: invalid 1: valid) when the 4th axis is the axis of rotation

035.1 When the 4th axis is the axis of rotation, (0: rotate in the direction of the symbol 1: rotate nearby).

035.2 Relative coordinate loop function when the 4th axis is the axis of rotation (0: invalid 1: valid).

035.3 When the Z axis is the axis of rotation (0: rotation in the direction of the symbol 1: rotation nearby).

035.4 The type when setting the X axis to the axis of rotation (0: axis of rotation A type 1: axis of rotation type B).

035.5 The type when setting the Y axis to the axis of rotation (0: axis of rotation A type 1: axis of rotation type B).

035.6 The type when setting the Z axis to the axis of rotation (0: axis of rotation A type 1: axis of rotation type B).

No. 36

036.1 The type when setting the 5th axis as the axis of rotation (0: axis of rotation A type 1: axis of rotation type B).

036.2 When the 6TH axis is the axis of rotation (0: rotate in the direction of the symbol 1: rotate nearby).

036.3 The type when setting the 6th axis as the axis of rotation (0: axis of rotation A type 1: axis of rotation type B).

036.4 Whether the 6TH axis requires a coordinate system tool interface (0:No 1:Yes)

036.5 Cs axis function for 5th axis (0: invalid 1: valid).

036.6 Whether the spindle impulses in direction (0:No 1:Yes) The direction of direction is positive

No. 37

037.0 Absolute coordinate loop function (0: invalid 1: valid) when the 5th axis is the axis of rotation

037.1 When the 5th axis is the axis of rotation, (0: rotate in the direction of the symbol 1: rotate nearby).

037.2 Relative coordinate loop function when the 5th axis is the axis of rotation (0: invalid 1: valid).

037.6 Rotate 0 bits of negative angle type by sign (00:Type A 01:Type B 10:Type C 11:Type D).

037.7 Rotate negative angle type 1 bit by sign (00:Type A 01:Type B 10:Type C 11:Type D).

No. 38

038.0 X-axis reads absolute value coordinates (0: do not negate 1: negation).

038.1 Y-axis reads absolute value coordinates (0: do not negate 1: negation).

038.2 Z-axis reads absolute value coordinates (0: do not negate 1: negation).

038.3 4TH axis read absolute value coordinates (0: do not negate 1: negation).

038.4 5TH axis read absolute value coordinates (0: do not negate 1: negation).

038.5 6TH axis read absolute value coordinates (0: do not negate 1: negation).

038.6 Spindle encoder feedback direction (0: do not negate 1: negation).

038.7 Whether the rotation axis updates the coordinate zero point in real time when reading the absolute value (0:invalid 1:valid)).

No. 39

039. 0 X-axis reads the absolute encoder number (0:17 bits 1:23 bits).

039.1 Y-axis reads absolute encoder bits (0:17 bits 1:23 bits).

039.2 Z-axis reads absolute encoder bits (0:17 bits 1:23 bits).

039.3 4-axis read absolute encoder bits (0:17 bits 1:23 bits).

039.4 5-axis read absolute encoder bits (0:17 bits 1:23 bits).

039.5 6-axis read absolute encoder bits (0:17 bits 1:23 bits).

Number 42

042. 0 Whether to use constant surface cutting speed control (0:No 1:Yes)

042.1 G200-G205 follow function (0: invalid 1: valid).

042.3 Rotation angle of G68 coordinate rotation (0: absolute command 1: G90/G91 command).

042.7 The Z axis moves in (0:G00 1:G01) during program restart

Number 43

043. 0 Whether to use the zoom function (0:No 1:Yes).

043.2 Whether X-axis scaling is valid (0:No 1:Yes).

043.3 Whether Y-axis scaling is valid (0:No 1:Yes).

043.4 Whether Z-axis scaling is valid (0:no 1:Yes).

043.7 Specify the scale ratio of each axis (0: P command for each axis 1: IJK command for each axis).

Number 44

044. 3 G76, G87 displacement (0:Q command 1:I,J,K command)

044.4 Set the G76, G87 retract direction (0:positive 1:negative).

044.5 Set the G76, G87 tool retract axis (0:X axis 1:Y axis).

044.7 Deep hole drilling (G73, G83), when the cutting amount is not commanded (0: no alarm 1: alarm).

Number 45

045.0 When reset or emergency stop, whether to clear the F, H, D code (0:No 1:Yes)

045.1 (0:No 1:Yes) Wait for the end of the M or S code before changing the analog voltage output

045.2 Whether to activate the multi-spindle control function (0:no 1:Yes).

Number 46

046.1 Whether to clear the 01 Group G code (0:No 1:Yes) when reset or emergency stop

046.2 Whether to clear the 02 Group G code (0:No 1:Yes) when reset or emergency stop

046.3 Whether to clear the 03 group G code (0:No 1:Yes) when reset or emergency stop

046.4 Clear Group 04 G code (0:No 1:Yes) when reset or emergency stop

046.5 Whether to clear the 05 group G code (0:No 1:Yes) when reset or emergency stop

046.6 Whether to clear the 06 group G code (0:No 1:Yes) when reset or emergency stop

046.7 Whether to clear the 07 Group G code (0:No 1:Yes) when reset or emergency stop

No. 47

047. 0 Whether to clear the 08 group G code when reset or emergency stop (0:No 1:Yes)

047.1 Whether to clear the 09 group G code (0:No 1:Yes)) when reset or emergency stop

047.2 Whether to clear 10 groups of G-codes (0:No 1:Yes) when reset or emergency stop

047.3 Whether to clear 11 G-codes (0:No 1:Yes) when reset or emergency stop

047.4 Whether to clear 12 groups of G codes (0:no 1:Yes) when reset or emergency stop

047.5 Whether to clear 13 groups of G-codes (0:No 1:Yes) when reset or emergency stop

047.6 Clear 14 G-codes (0:No 1:Yes) when reset or emergency stop

047.7 Clear 15 groups of G-codes (0:No 1:Yes) on reset or emergency stop

Number 48

048.0 When the power is turned on or the state is cleared (0:G00 mode 1:G01 mode).

048.1 Whether the plane selection is G17 when power is turned on or cleared (0:no 1:Yes).

048.2 Whether the plane selection is G18 when power is on or clear state (0:No 1:Yes).

048.3 Whether the plane selection is G19 when power is turned on or the state is cleared (0:No 1:Yes).

048.4 Set when power is turned on or clear state (0:G90 mode, 1:G91 mode).

048.5 Set when power is turned on or when state is cleared (0:G12,1:G13).

Number 49

049. 0 Whether it is forbidden to enter the tool wear offset amount via MDI (0: no 1: Yes).

049.1 Whether it is forbidden to enter the amount of tool geometric offset via MDI (0:no 1:Yes).

049.2 Whether to prohibit macro program variables from being entered via MDI (0:No 1:Yes).

049.3 Whether to prohibit input of workpiece origin bias via MDI (0:No 1:Yes)

No. 50

050.0 Whether to automatically insert sequential numbers (0:No 1:Yes).

050.1 When the program status interface is executed, the starting behavior (0: first line 1: the line where the cursor is located).

050.2 MDI interface after executing the program (0:No 1:Yes) delete the program

050.3 Press the reset key (0:No 1:Yes) on the MDI interface to delete the program

050.7 Absolute position display (0: not considered 1: considered) Tool length compensation

Number 51

051.0 Stroke detection before moving (0:No 1:Yes)

051.1 Arc interpolation travel detection (0:no 1:Yes).

051.2 Fast Moving Soft Limit Alarm (0:Hysteresis 1: Accurate)

051.4 M3 communicates the last address as a bus spindle function (0: invalid 1: valid).

051.5 M3 bus spindle speed feedback (0: external encoder 1: M3 bus feedback).

051.7 All shaft interlock signals are valid (0:no 1:Yes).

No. 52

052.2 Whether the one-way positioning G-code is set to modal code (0:no 1:yes)

052.3 Whether the indexing command and other control axis commands are alarmed in the same section (0:No1:Yes)

052.6 Is it possible to use a single segment (0:no 1:yes)) in macro program instruction statements

052.7 Whether there is delay in macro program instruction statements (0:delay 1:No delay)

No. 53

053. 6 Relative coordinate system after reset (0: do not cancel 1: cancel).

053.7 Zero relative coordinates (0:no 1:Yes) canceled

No. 54

054.0 When no position encoder is installed, the feed feed (0: invalid, 1: valid).

054.1 Whether the system has a position encoder installed (0:none, 1:Yes).

No. 55

055. 0 Whether the cutting feed is controlled to the accuracy in place (0:no 1:Yes)

055.1 Whether the automatic corner multiplier function is effective (0:no 1:Yes).

055.2 Whether the exponential acceleration acceleration of acceleration and deceleration cutting feed is clamped (0:no 1:Yes)

No. 56

056.3 (0:No 1:Yes) Use the handwheel compensation function

056. 5 Cut feed empty run (0: invalid, 1: valid).

056.6 Quickly locate whether an empty run is valid (0: invalid, 1: valid).

056.7 Empty run during tapping (0: invalid 1: valid).

No. 57

057.4 G0 is positioned when calculating the G96 spindle speed according to (0: end coordinate 1: current coordinate).

057.5 G96 spindle speed clamping (0: before spindle magnification 1: after spindle magnification).

057.7 In per-turn feed mode, whether G04 is paused per revolution (0:No 1:Yes)?

No. 58

058.0 Spindle (0:Follow 1:Servo) when tapping

058.1 Whether the tapping becomes a high-speed deep hole tapping cycle (0:no 1:Yes).

058.2 Z-axis direction selection negation (0: invalid, 1: valid) when rigid servo tapping

058.3 Whether the spindle is quasi-stop (0:no 1:yes) at the beginning of flexible tapping

058.4 Rigid tapping approach, whether the retract uses the same time constant (0:no 1:yes)

058.6 Whether the magnification is valid when rigid tapping and retracting (0:no 1:Yes).

No. 59

059.0 Fast running magnification adjustment use (0: operator panel button 1: band switch).

059.1 Cutting feed rate adjustment use (0: operator panel button 1: band switch).

059.2 Spindle speed multiplier adjustment (0: operator panel button 1: band switch).

059.3 PLC Level 1 program signals (0:no 1:Yes) use fast signals

059.4 Whether to use the handwheel/step interrupt function (0:no 1:Yes).

059.5 Whether to use an external handwheel (0:No 1:Yes)

059.6 Whether to use an external operator panel lock (0:No 1:Yes).

059.7 Whether to use an external edit lock (0:no 1:Yes).

Number 60

060. 0 Whether to ignore feedshaft drive alarms (0:No 1:Yes).

060.1 Whether to ignore spindle drive alarms (0:No 1:Yes).

060.2 Whether to ignore emergency stop alarms (0:no 1:Yes).

060.3 Whether to ignore hard limit alarms (0:no 1:Yes).

060.4 Whether to ignore external user alarms (0:No 1:Yes).

060.5 Whether to shield the bus connection alarm (0:No 1:Yes).

060.6 Whether the slave alarm is automatically canceled (0:no 1:Yes).

Number parameters:

000 X-axis instruction multiplication factor (numerator)

001 Y-axis command frequency doubling coefficient (numerator)

002 Z-axis command frequency doubling coefficient (numerator)

003 4TH axis command frequency doubling coefficient (numerator)

004 5TH Axis Command Doubling Coefficient (Numerator)

005 X-axis command divider factor (denominator) (when directly connected, screw pitch setting)

006 Y-axis command frequency division factor (denominator) (when directly connected, screw pitch setting)

007 Z-axis command frequency division factor (denominator) (when directly connected, screw pitch setting)

008 4TH axis command divider factor (denominator) (when directly connected, screw pitch setting)

009 5TH axis command frequency division factor (denominator) (when directly connected, screw pitch setting)

010 X-axis soft limit negative maximum stroke setting (record position by axis number and input)

011 X-axis soft limit forward maximum stroke setting (record position by axis number and input)

012 Y-axis soft limit negative maximum stroke setting (record position by axis number and input)

013 Y-axis soft limit forward maximum stroke setting (record position by axis number and input)

014 Z-axis soft limit negative maximum stroke setting (record position by axis number and input)

015 Z-axis soft limit forward maximum stroke setting (record position by axis number and input)

016 4TH Axis Soft Limit Negative Maximum Travel Setting (Record Position by Axis Number and Input)

017 4TH Axis Soft Limit Forward Maximum Stroke Setting (Record position by axis number by input)

018 5TH Axis Soft Limit Negative Maximum Stroke Setting (Record position by axis number by input)

019 5TH Axis Soft Limit Forward Maximum Stroke Setting (Record position by axis number by input)

020 X-axis negative maximum stroke (second stroke limit)

021 X-axis positive maximum stroke (second stroke limit)

022 Y-axis negative maximum stroke (second stroke limit)

023 Y-axis positive maximum stroke (second stroke limit)

024 Z-axis negative maximum stroke (second stroke limit)

025 Z-axis positive maximum stroke (second stroke limit)

026 4TH Axis Negative Maximum Travel (Second Stroke Limit)

027 4TH Axis Forward Maximum Travel (Second Stroke Limit)

028 5TH Axis Negative Maximum Travel (2nd Stroke Limit)

029 5TH Axis Positive Maximum Travel (2nd Stroke Limit)

030 Accuracy of Backlash Compensation to Determine Reverse Direction (X0.0001)

031 The amount of backlash compensation for the X-axis

032 Backlash compensation for the Y-axis

033 Backlash compensation for the Z-axis

034 4TH axis backlash compensation amount

035 5TH axis backlash compensation amount

The 036 X-axis gap compensates step size at a fixed frequency

The 037 Y-axis gap compensates step size at a fixed frequency

The 038 Z-axis gap compensates for step size at a fixed frequency

039 4-axis clearance compensates step size at a fixed frequency

040 5TH AXIS CLEARANCE COMPENSATED IN FIXED FREQUENCY COMPENSATED STEPS

041 Backlash rate compensation time constant

042 X-axis reverse positioning tolerance

043 Y-axis reverse positioning tolerance

044 Z-axis reverse positioning tolerance

045 4TH axis reverse positioning tolerance

046 5TH AXIS REVERSE POSITIONING TOLERANCE

047 2nd pulse spindle maximum speed

048 When the analog voltage output of the second spindle is 10V, the corresponding maximum speed

049 ladder step setting

050 X-axis value at the first reference point in the mechanical coordinate system (record position by axis number and input)

051 Y-axis value at the first reference point of the mechanical coordinate system (record position by axis number and input)

052 Z-axis value at the first reference point of the mechanical coordinate system (record position by axis number and input)

053 4TH axis in the mechanical coordinate system 1st reference point value (record position by axis number and input)

054 5TH axis in the mechanical coordinate system 1st reference point value (record position by axis number and input)

055 X-axis value at the second reference point in the mechanical coordinate system (record position by axis number and input)

056 Y-axis value at the second reference point of the mechanical coordinate system (record position by axis number and input)

057 Z-axis at the second reference point value of the mechanical coordinate system (tool change downward loading point)

058 4TH axis in the mechanical coordinate system 2nd reference point value (record position by axis number and input)

059 5TH axis in the mechanical coordinate system 2nd reference point value (record position by axis number and input)

060 X-axis at the 3rd reference point value of the mechanical coordinate system (record position by axis number and input)

061 Y-axis value at the 3rd reference point of the mechanical coordinate system (record position by axis number and input)

062 Z-axis at the 3rd reference point value of the mechanical coordinate system (loosening knife upward point)

063 4TH axis in the mechanical coordinate system 3rd reference point value (record position by axis number and input)

064 5TH axis in the mechanical coordinate system 3rd reference point value (record position by axis number and input)

065 X-axis value at the 4th reference point of the mechanical coordinate system (record position by axis number and input)

066 Y-axis value at the 4th reference point of the mechanical coordinate system (record position by axis number and input)

067 Z-axis at the 4th reference point value of the mechanical coordinate system (record position by axis number and input)

068 4TH 4TH REFERENCE POINT VALUE IN THE MECHANICAL COORDINATE SYSTEM (RECORD POSITION BY AXIS NUMBER AND INPUT)

069 5TH 4th reference point value in the mechanical coordinate system (record position by axis number and input)

070 X-axis returns to the high speed of the mechanical zero point

071 The Y-axis returns to the high-speed speed of the mechanical zero point

072 The Z-axis returns to the high speed of the mechanical zero point

073 4TH AXIS RETURNS TO THE HIGH SPEED OF MECHANICAL ZERO

074 5TH AXIS RETURNS TO MECHANICAL ZERO AT HIGH SPEED

075 The grid offset or reference point offset for the X-axis

The grid offset or reference point offset for the 076 Y-axis

077 The grid offset or reference point offset for the Z-axis

078 Grid offset or reference point offset for the 4TH axis

079 5TH AXIS GRID OFFSET OR REFERENCE POINT OFFSET

080 The low-speed speed at which the full shaft returns to the mechanical zero point

081 Zero detection range mm

082 Empty running speed

083 Cutting feed rate when power is turned on

084 Cutting feed internal given speed

085 Fo speed at fast running magnification of each axis (common to all axes)

086 Fast positioning and non-read-ahead feed maximum control speed (universal to all axes)

087 Fast positioning and non-read-ahead feeding minimum control speed (universal for all axes)

088 Maximum control speed in read-ahead mode (universal for all axes)

089 Minimum control speed in read-ahead mode (universal for all axes)

090 X-axis G0 fast positioning speed

091 Y-axis G0 fast positioning speed

092 Z-axis G0 fast positioning speed

093 4TH axis G0 fast positioning speed

094 5TH axis G0 fast positioning speed

095 6TH axis G0 fast positioning speed

096 7TH axis pulse spindle molecule

097 7TH axis pulse spindle denominator

098 7TH Fast Forward Acceleration and Deceleration Linear Type/Pulse Spindle Acceleration and Deceleration Time Constant

099 Panel model code

100 exponential acceleration acceleration clamping constant

101 Reserved

102 handwheel incomplete running mode maximum clamping speed

103 Handwheel incomplete running mode acceleration clamping constant

104 Handwheel linear acceleration and deceleration time constant

105 Handwheel exponential acceleration and deceleration time constant

106 Arc deceleration radius 1 corresponding speed (default: 2000)

107 Arc deceleration radius 2 corresponding speed (default: 2500)

108 single step for maximum clamping speed

109 Arc deceleration radius 3 corresponding speed (default: 3000)

110 Continuous feed speed per axis manual (JOG).

111 Linear acceleration and deceleration time constant for manual feeding of each axis

112 Exponential acceleration and deceleration time constant for manual feed of each axis

113 X-axis manual fast positioning speed

114 Y-axis manual fast positioning speed

115 Z-axis manual fast positioning speed

116 4TH axis manual fast positioning speed

117 5TH AXIS MANUAL FAST POSITIONING SPEED

118 Arc speed relative to straight-line speed deceleration percentage setting

119 Arc Max Programmed Speed Limit (default: 5000)

120 Linear time constant for rapid X-axis front acceleration and deceleration

121 Linear time constant for rapid Y-axis forward acceleration and deceleration

122 Linear time constant for rapid Z-axis forward acceleration and deceleration

123 Fast 4th axis front acceleration and deceleration linear type/pulse spindle acceleration and deceleration time constant

124 Fast 5th axis front acceleration and deceleration linear type/pulse spindle acceleration and deceleration time constant

125 S-type time constant for rapid X-axis front acceleration and deceleration

126 S-type time constant for rapid Y-axis front acceleration and deceleration

127 S-type time constant for rapid Z-axis front acceleration and deceleration

128 Fast 4Th axis front acceleration and deceleration S-type time constant

129 Fast 5Th axis front acceleration and deceleration S-type time constant

130 Linear time constant for acceleration and deceleration after fast X-axis

131 Linear time constant for acceleration and deceleration after fast Y-axis

132 Linear time constant for acceleration and deceleration after fast Z-axis

133 Linear time constant for acceleration and deceleration after fast 4th axis

134 Fast 5th axis post-acceleration and deceleration linear time constant

135 Exponential time constant for acceleration and deceleration after fast X-axis

136 Exponential time constant for acceleration and deceleration after fast Y-axis

137 Exponential time constant for acceleration and deceleration after the fast Z-axis

138 Exponential time constant for acceleration and deceleration after fast 4th axis

139 Exponential time constant for acceleration and deceleration after fast 5th axis

140 Non-read-ahead mode, maximum number of merged program segments

141 Non-read-ahead mode, merge program segment control accuracy

142 Non-read-ahead mode, cutting feed accuracy in place

143 Reserved

144 Acceleration and deceleration time constant before zeroing/non-read-ahead mode, cutting straight line time constant

145 Handwheel test cut reverse acceleration/non-pre-read mode, S-type time constant for acceleration and deceleration before cutting feed

146 Spindle pulse directional acceleration and deceleration time constant/non-pre-reading mode, linear time constant of acceleration and deceleration after cutting feed

147 Non-read-ahead mode, exponential time constant of acceleration and deceleration after cutting feed

148 Non-read-ahead mode, exponential acceleration and deceleration minimum speed (FL)

149 handwheel test cut maximum speed limit

150 read-ahead mode, maximum number of merge program segments

151 Pre-read mode, merge program segment control accuracy

152 pre-read mode, cutting feed accuracy in place

153 Hand wheel test cut follows the base of change

154 Read-ahead method, acceleration of linear type acceleration (mm/s/s) before cutting feed

155 Read-ahead mode, S-type time constant for acceleration and deceleration before cutting feed

156 Read-ahead mode, linear acceleration and deceleration time constant after cutting feed

157 Read-ahead mode, exponential acceleration and deceleration time constant after cutting feed

158 Read-ahead mode, exponential acceleration and deceleration minimum speed (FL)

159 Link mode system coordinates and motor coordinate alarm difference

160 Pre-read mode, the critical angle (degree) of the two program segments of automatic corner deceleration.

161 pre-read mode, automatic corner deceleration, minimum feed speed

162 Pre-read mode, speed difference mode deceleration function allowable deviation of each axis

163 Read-ahead mode, cutting accuracy level

164 Reserved

165 Pre-read mode, which constitutes the spline length condition

166 Pre-read mode, which constitutes the spline angle condition

167 Read-ahead mode, cutting feed acceleration and deceleration smoothing processing time constant

168 Read-ahead mode, cutting feed, acceleration and deceleration post-processing time constant

169 Reserved

170 arc interpolation external acceleration limit

171 Arc interpolation of the lower limit of the low speed of the external acceleration embedding

172 arc interpolation control accuracy

173 Arc radius error limit

174 Feed shaft as pulse spindle stop minimum speed (default 1)

175 Model S acceleration plus acceleration maximum

176 S-type deceleration acceleration maximum

177 Arc reduction radius 1 (minimum) (default: 5)

178 Arc reduction radius2 (default: 8)

179 Arc reduction radius 3 (maximum) (default: 10)

Pitch error compensation number for 180 X-axis reference point

Pitch error compensation number for 181 Y-axis reference point

Pitch error compensation number for 182 Z-axis reference point

183 4TH AXIS REFERENCE POINT PITCH ERROR COMPENSATION NUMBER

184 5TH AXIS REFERENCE POINT PITCH ERROR COMPENSATION NUMBER

185 X-axis pitch error compensation points

186 Y-axis pitch error compensation points

187 Z-axis pitch error compensation points

188 4TH shaft pitch error compensation points

189 5TH shaft pitch error compensation points

190 X-axis pitch error compensation spacing

191 Y-axis pitch error compensation spacing

192 Z-axis pitch error compensation spacing

193 4TH shaft pitch error compensation spacing

194 5TH SHAFT PITCH ERROR COMPENSATION SPACING

195 X-axis pitch error compensation magnification

196 Y-axis pitch error compensation magnification

197 Z-axis pitch error compensation magnification

198 4TH shaft pitch error compensation magnification

199 5TH shaft pitch error compensation magnification

200 spindle maximum speed

201 Number of spindle encoder lines

The maximum setting value for the 202 frequency converter

203 Lower limit value of spindle magnification

204 Reserved

Gain adjustment data for 205 spindle speed analog output

206 Spindle speed analog output bias voltage compensation value

207 Current cutting time

208 Spindle speed when the spindle is oriented or jogged

209 Upper spindle speed during tapping cycles

210 corresponds to the maximum spindle speed of gear 1

211 corresponds to the maximum spindle speed of gear 2

212 corresponds to the maximum spindle speed of gear 3

213 corresponds to the maximum spindle speed of gear 4

214 Output voltage (mV) when spindle shifts

215 Stepped acceleration and deceleration time constant for each axis JOG feed

216 Z-axis tool change slow range limits speed F

217 Z-axis height of the workpiece origin (0 time back to mechanical zero)

218 X-axis tool change slow zone limit speed F

219 Y-axis tool change slow range limit speed F

220 Number of teeth of spindle side gears (1st gear)

221 Number of teeth of spindle side gears (2nd gear)

222 Number of teeth of spindle side gears (3rd gear)

223 Number of gear teeth on the side of the position encoder (1st gear)

224 Number of gear teeth on the side of the position encoder (2nd gear)

225 Position encoder side gear teeth (3rd gear)

The 226 Turret changer allows a minimum rotation interval of F23.4

The maximum allowable rotation range of the 227 turret changer

228 Z-axis tool change slow range minimum F23.5

229 Z-axis tool change slow range maximum

230 Tapping Spindle Command Multiplier Factor (CMR) (spindle motor line speed setting value)

231 Tapping Spindle Command Multiplier Factor (CMR) (2nd gear gear)

232 Tapping Spindle Command Multiplier Factor (CMR) (3rd gear gear)

233 Tapping Spindle Command Divider Factor (CMD) (default: 250)

234 Tapping Spindle Command Divider Factor (CMD) (2nd gear gear)

235 Tapping Spindle Command Divider Factor (CMD) (3rd gear gear)

236 Turret tool magazine spindle forbidden interval minimum F23.6

237 Turret changer spindle forbidden range maximum

238 Turret changer minimum F23.7

239 Turret changer maximum tool change interval

240 Clearance of rigid tapping spindle (1st gear gear)

241 Clearance of rigid tapping spindle (2nd gear gear)

242 Clearance of rigid tapping spindle (3rd gear)

243 High-speed machining corner speed (default: 300)

244 high-speed machining acceleration (default: 500)

245 High Speed Machining Efficiency Level (default: 0)

246 high-speed machining optimizations (default: 10)

247 abs. (0:KY1000 1:SANKYO 2:98D)

248 absolute stop bit (0: odd check 1: no check)

249 Reserved

250 Linear acceleration and deceleration time constant for spindle and tapping shaft (1st gear gear)

251 Linear acceleration and deceleration time constant for spindle and tapping shaft (2nd gear gear)

252 Linear acceleration and deceleration time constant for spindle and tapping shaft (3rd gear)

253 Linear acceleration and deceleration time constant of spindle and tapping shaft during tool retraction (1st gear gear)

254 Linear acceleration and deceleration time constant of spindle and tapping shaft during tool retraction (2nd gear gear)

255 Linear acceleration and deceleration time constant of spindle and tapping shaft during tool retraction (3rd gear gear)

256 The amount of compensation of the tapping shaft during rigid tapping

257 Reserved

258 Number of backlash direction checks (default 5 times)

259 pulse direction completion delay

260 External workpiece origin X-axis offset

261 External workpiece origin Y-axis offset

262 Z-axis offset of the external workpiece origin

263 EXTERNAL WORKPIECE ORIGIN 4TH AXIS OFFSET

264 EXTERNAL WORKPIECE ORIGIN 5TH AXIS OFFSET

265 The workpiece origin offset of the workpiece coordinate system 1 (G54_X).

266 The workpiece origin offset of the workpiece coordinate system 1 (G54_Y).

267 The workpiece origin offset of workpiece coordinate system 1 (G54_Z).

268 The workpiece origin offset of the workpiece coordinate system 1 (G54_4TH).

269 The workpiece origin offset of the workpiece coordinate system 1 (G54_5TH).

270 Workpiece origin offset for workpiece coordinate system 2 (G55_X).

271 The workpiece origin offset of the workpiece coordinate system 2 (G55_Y).

272 The workpiece origin offset of the workpiece coordinate system 2 (G55_Z).

273 Workpiece origin offset of workpiece coordinate system 2 (G55_4TH).

274 The workpiece origin offset of the workpiece coordinate system 2 (G55_5TH).

275 Workpiece origin offset of workpiece coordinate system 3 (G56_X).

276 The workpiece origin offset of the workpiece coordinate system 3 (G56_Y).

277 The workpiece origin offset of the workpiece coordinate system 3 (G56_Z).

278 The workpiece origin offset of the workpiece coordinate system 3 (G56_4TH).

279 The workpiece origin offset of the workpiece coordinate system 3 (G56_5TH).

280 Workpiece origin offset of workpiece coordinate system 4 (G57_X).

281 The workpiece origin offset of the workpiece coordinate system 4 (G57_Y).

282 Workpiece origin offset for workpiece coordinate system 4 (G57_Z).

283 The workpiece origin offset of the workpiece coordinate system 4 (G57_4TH).

284 The workpiece origin offset of the workpiece coordinate system 4 (G57_5TH).

285 The workpiece origin offset of the workpiece coordinate system 5 (G58_X).

286 The workpiece origin offset of the workpiece coordinate system 5 (G58_Y).

287 The workpiece origin offset of the workpiece coordinate system 5 (G58_Z).

288 The workpiece origin offset of the workpiece coordinate system 5 (G58_4TH).

289 The workpiece origin offset of the workpiece coordinate system 5 (G58_5TH).

290 The workpiece origin offset of the workpiece coordinate system 6 (G59_X).

291 Workpiece origin offset of workpiece coordinate system 6 (G59_Y).

292 The workpiece origin offset of the workpiece coordinate system 6 (G59_Z).

293 The workpiece origin offset of the workpiece coordinate system 6 (G59_4TH).

294 The workpiece origin offset of the workpiece coordinate system 6 (G59_5TH).

295 Rigid tapping Z-axis compensation error

296 Rigid tapping spindle compensates for errors

297 SPINDLE DIRECTIONAL AXIS SETUP (4:4TH 5 OR OTHER: 5TH 6:6TH 7:7TH)

298 spindle directional speed

299 Spindle orientation angle

300 DNC (0:USB stick 1:Xon 2:XModem)

301 Communication Channel Baud Rate (DNC)

302 communication channel baud rate (transfer file)

303 CNC controls the number of axes

304 The ladder number currently in use

305 System Language Selection (0:CH 1:EN 2:RUS 3:ESP)

306 Spindle pulse direction allowable range (default 6)

307 splash screen background color (0:black 1:white 2:red 3:green 4:blue)

308 Progress bar color at startup (0:yellow 1:white 2:red 3:green 4:blue)

309 The number of pages of user parameters above the second-level password operation level that are available for user modification

310 Program name for axis 1

311 Program name for axis 2

312 Program name for axis 3

313 Program names for axis 4 (3:A,4:B,5:C)

314 Program names for axis 5 (3:A,4:B,5:C)

315 pair of knife speeds

316 The increment value of the number of sequential numbers when the sequence number is automatically inserted

317 Prohibit MDI from entering the beginning number of the tool offset amount

318 Prohibit MDI from entering the number of tool offsets

319 Reserved

320 Reserved

Output time of the 321 reset signal

The number of allowed bits for the 322 M code

The number of allowed bits for the 323 S code

The number of allowed digits for the 324 T code

325 MST code colors (0:black 1:red2violet 3:blue 4:light blue 5:cyan 6:yellow 7:green)

326 Spindle Pulse Port Selection (4:4th 5 or Other: 5th 6:6th 7:7th)

327 The axis that serves as the counting reference when the surface speed is controlled

Minimum spindle speed at 328 constant surface speed control (G96).

329 G23/G24/G33/G34 spiral downward speed

330 Tool radius compensation C ignores the limit value of the vector when moving along the outside of the corner

331 Maximum amount of tool wear compensation

332 Tool radius compensates for the maximum error value of C

Coefficient of the radius of the spiral down knife in the 333 groove cycle

334 Retract volume of high-speed deep hole cycle G73 (default: 2)

335 Empty amount for fixed cycle G83 (default: 2)

Minimum pause time at the bottom of 336 wells

Maximum pause time at the bottom of 337 wells

338 Magnification value when rigid tapping and retracting

339 Amount of backoff or leave blank during deep hole tapping cycles

340 days of early alarm when the encryption function expires

341 Rigid tapping retraction speed

The rotation angle used when there is no rotation angle command in the 342 G68 coordinate rotation

343 Encryption function machine number (this function is not valid when it is 0)

344 Zoom factor used when there is no zoom instruction

345 X-axis zoom magnification

Scale magnification of the 346 Y-axis

Scale magnification of the 347 Z axis

348 Reserved

349 Reserved

350 pause time when positioning in one direction

351 X-axis unidirectional positioning direction and overtravel

352 Y-axis unidirectional positioning direction and overtravel

353 Z-axis unidirectional positioning direction and overtravel

354 4TH axis unidirectional positioning direction and overtravel

355 5TH axis unidirectional positioning direction and overtravel

356 6TH axis unidirectional positioning direction and overtravel

357 Reserved

358 The sequence of movement of each axis to the program start position

359 Clear F signal wait time after key release (original default 50)

360 machined pieces

361 pieces in demand

362 Cumulative value of power-on time (hours)

363 Cumulative value of cutting time (hours)

364 Reserved

365 interface color adjustment display

366 interface color adjustment display 2

367 X-axis tool change slow range minimum F23.2

368 X-axis tool change slow range maximum

369 Number of program lines displayed under the graphical interface

370 Program number of execution program segment (invalid if 0)

371 The line number of the start program in the execution program segment

372 End program line number of the execution program segment

373 Repeated tool setting error value

The 374 system powers on to set the spindle speed value

375 X-axis cutting feed for maximum control speed

376 Y-axis cutting feed maximum control speed

377 Z-axis cutting feed maximum control speed

378 4TH axis cutting feed for maximum control speed

379 5TH AXIS CUTTING FEED MAXIMUM CONTROL SPEED

380 Y-axis tool change slow interval minimum F23.3

381 Y-axis tool change slow range maximum

382 Zero detection reserved pulse number (default 10)

The 383 X-axis returns to zero and leaves the stop to detect a turn signal distance

The 384 Y-axis returns to zero and leaves the stop to detect a turn signal distance

The 385 Z-axis returns to zero and leaves the stop to detect a turn signal distance

386 4-axis zero off the stop to detect a turn signal distance

387 5-axis zero off the stop to detect a turn signal distance

388 6-axis zero exit stop to detect a turn signal distance

389 Reserved

390 Reserved

391 Reserved

392 Reserved

393 Reserved

394 Reserved

395 X-axis speed limit magnification setting

396 Y-axis speed limit magnification setting

397 Z-axis speed limit magnification setting

398 4TH AXIS SPEED LIMIT MAGNIFICATION SETTING

399 5TH axis speed limit magnification setting

400 tool x-axis position

401 Tool Setter Y-axis position

402 Z-axis tool starting point

403 measurement mode (0: single pole 1: multi-pole 2 reserved)

404 Reserved

405 Workpiece coordinate system

406 Setter Z-axis nadir point

407 Z-axis tool setter drop value

408 Positioning coordinate error alarm range when rowing knives

409 Reserved

410 Servo X-axis instruction multiplication factor (CMR)

411 Servo Y-axis instruction multiplication factor (CMR)

412 Servo Z-axis command multiplication factor (CMR)

413 Servo 4th axis instruction multiplication factor (CMR)

414 Servo 5th axis command multiplier factor (CMR)

415 Servo X-axis Command Divider Factor (CMD)

416 Servo Y-axis command frequency division factor (CMD)

417 Servo Z-axis command frequency division factor (CMD)

418 Servo 4th Axis Command Divider Factor (CMD)

419 Servo 5th Axis Command Divider Coefficient (CMD)

420 6TH Axis Command Doubling Coefficient (Numerator)

421 6TH AXIS COMMAND DIVIDER FACTOR (DENOMINATOR)

422 Servo 6th axis command multiplication factor (CMR)

423 Servo 6th axis command divider factor (CMD)

424 EXTERNAL WORKPIECE ORIGIN 6TH AXIS OFFSET

425 Workpiece origin offset for coordinate system 1 (G54_6TH).

426 Workpiece origin offset for workpiece coordinate system 2 (G55_6TH).

427 The workpiece origin offset of the workpiece coordinate system 3 (G56_6TH).

428 The workpiece origin offset of the workpiece coordinate system 4 (G57_6TH).

429 The workpiece origin offset of the workpiece coordinate system 5 (G58_6TH).

430 The workpiece origin offset of the workpiece coordinate system 6 (G59_6TH).

431 6TH 1st reference point value in the mechanical coordinate system (record position by axis number by input)

432 6TH 2nd reference point value in the mechanical coordinate system (record position by axis number and input)

433 6TH 3rd reference point value in the mechanical coordinate system (record position by axis number by input)

434 6TH 4TH REFERENCE POINT VALUE IN THE MECHANICAL COORDINATE SYSTEM (RECORD POSITION BY AXIS NUMBER AND INPUT)

435 6TH Axis Soft Limit Negative Maximum Travel Setting (Record Position by Axis Number by Input)

436 6TH Axis Soft Limit Forward Maximum Stroke Setting (Record Position by Axis Number by Input)

437 6TH Axis Negative Maximum Travel (Second Stroke Limit)

438 6TH Axis Positive Maximum Travel (Second Stroke Limit)

439 6TH AXIS AMOUNT OF BACKLASH COMPENSATION

440 6TH AXIS CLEARANCE COMPENSATED AT A FIXED FREQUENCY COMPENSATED STEP SIZE

441 6TH AXIS REVERSE POSITIONING TOLERANCE

442 6TH shaft fast moving front acceleration and deceleration linear type/pulse spindle acceleration and deceleration time constant

443 6TH AXIS FAST MOVING FRONT ACCELERATION AND DECELERATION S-TYPE TIME CONSTANT

444 6TH LINEAR TIME CONSTANT FOR ACCELERATION AND DECELERATION AFTER RAPID MOVEMENT

445 6TH AXIS ACCELERATION AND DECELERATION TYPE E-TYPE TIME CONSTANT AFTER RAPID MOVEMENT

446 6TH AXIS MANUAL FAST MOVING SPEED

447 6TH axis cutting feed for maximum control speed

448 6TH returns to the machine zero high-speed speed

449 6TH speed limit magnification setting

450 6TH GRID OFFSET OR REFERENCE POINT OFFSET

451 6TH reference point pitch error compensation number

452 6TH pitch error compensation points

453 6TH pitch error compensation magnification

454 6TH pitch error compensation spacing

PLC parameter KEEP

K0

K000.1 1/0: The ladder interface data is displayed as hexadecimal/decimal

K000.3 1/0: Vertical key input/horizontal key input

K000.7 1/0: PLC enters debugging mode/operating mode

K1

K001.7 Automatic indexing table (0:forbidden 1:allow).

K001.6 Output after the indexing table is in place (0: uninterrupted 1: disconnected).

K001.5 Indexing table in place signal (0: detect 1: do not detect).

K001.4 The indexing table is not clamped (0:forbidden 1:allowed) XYZ axis movement

K001.3 Tool changer with the tool changer function turned on (0: use 1: not used).

K001.2 Ladder debugging (0:forbidden 1:Allow).

K001.1 Tool magazine use (0: other tool changers 1: row of tools).

K001.0 (0:forbidden 1:allow) Tool changer use

K2

K002.0 Machine lock button (0:Yes 1:No) is valid

K002.1 The secondary lock key (0:Yes 1:No) is valid

K002.7 When the indexing table is clamped in place (0:no 1:yes), the 4th axis is broken

K3

K003.1 Spindle gear hold register

K003.2 Spindle gear hold register

K003.4 External handwheel axle selection (0: valid 1: invalid).

K003. 6 Handwheel magnification signal MP1 hold register

K003.7 Handwheel multiplier signal MP2 hold register

K4

K004.0 X-axis negative hard limit register

K004.1 X-axis forward hard limit register

K004.2 Y-axis negative hard limit register

K004.3 Y-axis forward hard limit register

K004.4 Z-axis negative hard limit register

K004.5 Z-axis forward hard limit register

K004.6 4th axis negative hard limit register

K004.7 4th axis forward hard limit register

K5

K005.6 5th axis negative hard limit register

K005.7 5th axis forward hard limit register

K006 feed rate hold register

K007 Fast magnification hold register

K008 Spindle magnification hold register

K009 operates with hold registers

K10

K010.7 Hard limit overtravel for each axis (0: invalid 1: valid).

K010.6 The hard limit overtravel signal of each axis is (0:normally open 1:normally closed).

K010.5 Overtravel release signal for each axis (0: invalid 1: valid).

K010.4 Spindle jog is valid at (0: manual, handwheel, zero return method 1: any way).

K010.3 feed rate (0: magnification can be adjusted 1: fixed at 100%)

K010.1 Program end spindle/lubrication/cooling output (0: off 1: hold).

K010.0 Program running status skip button (0: valid 1: invalid).

K13

K013.7 External tightening knife input signal selection (0: jog 1: normal).

K013.6 System panel looser button (0: valid 1: invalid).

K14

K014.7 (0: not using 1: using) Tool clamping device

K014.6 (0: Check 1: Do not check) Tool clamping/unwinding in place signal

K014.5 Tool loosening and spindle (0: interlock 1: non-interlock).

K014.4 When the protective door is open (0: close the spindle 1: do not close the spindle).

K014.3 The guard door signal is (0:normally open 1:normally closed).

K014.2 Guard door function (0: invalid 1: valid).

K014.1 The tool clamping signal is (0: normally open 1: normally closed).

K014.0 The tool release signal is (0:normally open 1:normally closed).

K15

K015.7 CS axis function (0: invalid 1: valid).

K015.6 Emergency stop, spindle profile control when reset (0: not off 1: off).

K015.5 Spindle forward rotation signal when spindle automatic gear shifting (0:no 1:Yes).

K015.4 Delayed disconnect output after spindle shift (0: invalid 1: valid).

K015.3 Spindle gear (0:No 1:Yes) Power-down memory

K015.2 Automatic shift 1/2 gear in place signal is (0:NO 1:NC) is valid

K015.1 (0: Do not check 1: Check) Shift in place signal

K015.0 Spindle automatic shift function (0: invalid 1: valid).

K16

K016.7 Handwheel single-step X1000 increment (0: valid 1: invalid).

K016.5 Opening or closing the handwheel test cut during machining (0:1:No.1) pause

K016.4 Turn handwheel trial cut on or off during automatic run (0:invalid 1:Valid)

K016.3 Alarm prompt for completion of the total number of processing pieces (0: invalid 1: valid).

K016.2 Automatic lubrication output on system power-up (0:invalid 1:valid)

K016.1 The lubrication detection signal is (0:normally open 1:normally closed).

K016.0 External feed rate switch (0: invalid 1: valid).

K17

K017.0 Automatic output of work lights when the system is powered on (0: invalid 1: valid).

K017.1 Toggles F0 at fast multiplier (0:No 1:Yes) with a feed rate of 0%.

The K017.2 Y4.1 signal output is (0: pulse enable 1: 1st spindle forward rotation).

K017.3 The alarm signal of the second spindle is (0: normally open 1: normally closed).

K017.5 External spindle magnification switch (0: invalid 1: valid).

K017.6 M05 detects spindle zero speed mark when spindle stops (0:no 1:Yes).

K017.7 2nd spindle function (0: invalid 1: valid).

K18

K018.3 1/0: Operation mode designation/power-down memory

K018.2 Operation mode setting MD4: 110: mechanical zeroing

K018.1 Operation mode setting MD2: 011: Edit 100: Handwheel 101: Manual

K018.0 Operation mode setting MD1: 000: Enter 001: Automatic 010: DNC

K019 Manual multiplier hold register

K20

K020.0 external cycle start switch is (0:normally open 1:normally closed).

K020.1 The external feed hold switch is (0:NO 1:NC).

K020.2 The insufficient pressure detection signal is (0:NO 1:NC).

K020.3 Key automatic tool setting function (0: invalid 1: valid).

K020.4 One Click G54 Function (0: invalid 1: Valid).

K020.5 (0: External processing blowing button 1: Guard door signal) is valid

K020.6 X0.6 signal (0: lubrication detection 1: external control table) is valid

K21

K021.1 spindle positioning completion detection signal is (0: normally open 1: normally closed).

K021.7 Turn on manual quick movement control (0: jog 1: long press).

K22

K022.0 The Z axis is not at the zero point position and the other axes (0:Yes 1:No) can return to zero

K022.2 6-axis deceleration signal switching signal is (0:normally closed 1:normally open).

K022.3 5-axis deceleration signal switching signal is (0: normally closed 1: normally open).

K022.4 X-axis deceleration signal switching signal is (0:normally closed 1:normally open).

The K022.5 Y-axis deceleration signal switching signal is (0:normally closed 1:normally open).

K022.6 Z-axis deceleration signal switching signal is (0:normally closed 1:normally open).

K022.7 4-axis deceleration signal switching signal is (0:normally closed 1:normally open).

K24

K024.0 X-axis hard limit overtravel (0: valid 1: invalid) = premise K010.7 is 1

K024.1 Y-axis hard limit overtravel (0: valid 1: invalid) = premise K010.7 is 1

K024.2 Z-axis hard limit overtravel (0: valid 1: invalid) = premise K010.7 is 1

K024.3 4th axis hard limit overtravel (0: valid 1: invalid) = premise K010.7 is 1

K024.4 5th axis hard limit overtravel (0: valid 1: invalid) = premise K010.7 is 1

K024.6 The tool setter overtravel signal is (0:normally closed 1:normally open).

The K024.7 G31 input signal is (0: normally closed 1: normally open).

.K25

K025.0 Emergency Stop (0:Yes 1:No) turns off X-axis enable

K025.1 Emergency Stop (0:Yes 1:No) turns off Y-axis enablement

K025.2 Emergency Stop (0:Yes 1:No) turns off Z-axis enable

K025.3 Emergency Stop (0:Yes 1:No) turns off 4-axis enable

K025.4 Emergency stop (0:Yes 1:No) turns off 5-axis enable

K025.5 Emergency stop (0:Yes 1:No) turns off 6-axis enable

K27

K027.7 (0:M89I7Custom 1:Macro output #1107)Output Y2.5 is valid

K027.6 (0:M89I6Custom 1:Macro output #1106)Output Y2.1 is valid

K027.5 (0:M89I5Custom 1:Macro output #1105) Output Y1.3 is valid

K027.4 (0:M89I4Custom 1:Macro output #1104) Output Y1.2 is valid

K027.3 (0:M89I3Custom 1:Macro output #1103)Output Y1.1 is valid

K027.2 (0:M89I2Custom 1:Macro output #1102) Output Y1.0 is valid

K027.1 (0:M89I1Custom 1:Macro output #1101) Output Y0.7 is valid

K027.0 (0:M89I0Custom 1:Macro output #1100)Output Y0.5 is valid

K28

K028.7 (0: 2nd spindle stop 1: M89I7/macro output #1107) Output Y2.5 is valid

K028.6 (0: Brake output 1: M89I6 custom/macro output #1101) Output Y2.1 is valid

K028.5 (0: spindle 4 gears 1: M89I5 custom/macro output #1106) Output Y1.3 is valid

K028.4 (0: spindle 3 gears 1: M89I4 custom/macro output #1105) Output Y1.2 is valid

K028.3 (0: spindle 2 gears 1: M89I3 custom/macro output #1104) Output Y1.1 is valid

K028.2 (0: spindle 1 stop 1: M89I2 custom/macro output #1103) Output Y1.0 is valid

K028.1 (0: Spindle brake 1: M89I1 custom/macro output #1102) Output Y0.7 is valid

K028.0 (0: spindle stop 1: M89I0 custom/macro output #1100) Output Y0.5 is valid

K29

K029.7 Disconnect the M89I7 output when reset (0:Yes 1:No).

K029.6 Disconnect the M89I6 output when reset (0:Yes 1:No).

K029.5 Disconnect the M89I5 output when reset (0:Yes 1:No).

K029.4 disconnects the M89I4 output during reset (0:Yes 1:No).

K029.3 disconnects the M89I3 output during reset (0:Yes 1:No).

K029.2 disconnects the M89I2 output during reset (0:Yes 1:No).

K029.1 disconnects the M89I1 output when reset (0:Yes 1:No).

K029.0 disconnects the M89I0 output when reset (0:Yes 1:No).

K30

The K030.1 tool changer (0:No 1:Yes) has a zero switch

K030.2 The tool changer zero switch is (0: normally open 1: normally closed).

K030.3 The tool magazine forward/reverse tool in place signal is (0:normally open 1:normally closed).

K030.4 The tool changer back/return signal is (0:normally open 1:normally closed).

K030.5 Tool magazine counting switching signal is (0:NO 1:NC).

K030.6 Disc magazine arm/turret type (0:no 1:Yes) enters debug mode

K030.7 Tool changer (0:No 1:Yes) enters debug mode

K31

K031.0 detects the clamping signal when there is no tool after tool clamping (0:Yes 1:No).

K031.1 Output after the tool changer is forward/reversed in place (0: uninterrupted 1: disconnected).

K031.2 Tool magazine output after back/back tool in place (0:continuously open 1:disconnect).

K031.5 disc ATC brake signal is (0:NO 1:NC).

K031.6 Disc ATC gripper signal is (0:normally open 1:normally closed).

K031.7 Disc ATC origin signal is (0:normally open 1:normally closed).

K32

K032.0 Spindle with knife detection (0: invalid 1: valid).

K032.1 The spindle tool detection switch signal is (0:NO 1:NC).

K032.2 The current cutterhead magazine has a cutter detection (0: invalid 1: valid).

K032.3 The current cutterhead detection switch signal is (0:NO 1:NC).

K032.4 Shielding soft limit during tool change (0:no 1:Yes).

K032.5 Spindle alerts when the current tool number is 0 (0:Yes1:No).

K032.6 Tool magazine cutterhead rotation (0:no 1:Yes) has only forward rotation

K032.7 Manual/handwheel moves the Z-axis into the tool change zone (0:no 1:Yes) to trigger positioning

K34

K034.0 M89K0-K7/macro #1007-#1015 Detection signal selection (0:X3 1:X2)

K35

K035.0 M89K0/macro #1008 heartbeat (0:NO 1:NC).

K035.1 M89K1/Macro #1009 heartbeat (0:NO 1:NC).

K035.2 M89K2/Macro #1010 heartbeat (0:NO 1:NC).

K035.3 M89K3/Macro #1011 heartbeat (0: normally open 1: normally closed).

K035.4 M89K4/macro #1012 heartbeat (0:NO 1:NC).

K035.5 M89K5/macro #1013 heartbeat (0: normally open 1: normally closed).

K035.6 M89K6/macro #1014 heartbeat (0:NO 1:NC).

K035.7 M89K7/macro #1015 heartbeat (0:NO 1:NC).

K36

K036.0 Reset button (0:No 1:Yes) disconnects the guard gate output M34

K036.1 Reset button (0:No 1:Yes) disconnects the chip conveyor output M36

K036.2 Reset button (0:No 1:Yes) Disconnect the work lamp output M38

K036.3 Reset button (0:No 1:Yes) Disconnect the blowing output M70 for the knife

K036.4 Reset button (0:No 1:Yes) disconnects K2 output M72

K036.5 Reset button (0:No 1:Yes) disconnects K3 output M74

K036.6 Reset button (0:No 1:Yes) disconnects output M76

K50

K050.4 Turret type tool magazine Z-axis into the spindle no rotation zone sign

K050.5 Z-axis 1st reference

K050.6 Z-axis 2nd reference

K050.7 Turret type changer Z-axis entry tool magazine rotatable zone mark

K51

K051.1 The turret changer is not positioned with the Z-axis negative entry change zone flag

K051.2 The turret tool changer is not positioned with the Z-axis positive entry change zone flag

PLC parameter DATA

D100 total tool number setting

D101 command knife number

D102 The knife number before the command knife number

D103 Tool changer counts cutterhead positions

D104 Manual tool changer number

D105 spindle current tool number

D106 Disc Tool Change Data Sheet Rearrangement

D107 cutterhead number display

D120 Tool Changer Return/Finder Retrieval Address

D121 Tool Changer Return/Finder retrieves output results

D124 Tool magazine data table search range

D126 spindle zero speed output range (r/min)

D000 cutterhead No. 0 pocket setting

D001 Cutterhead No. 1 pocket setting

D002 Cutterhead No. 2 socket setting

D003 Cutterhead No. 3 socket setting

D004 Cutterhead No. 4 socket setting

D005 Cutterhead No. 5 socket setting
D006 Cutterhead No. 6 socket setting
D007 Cutterhead No. 7 socket setting
D008 Cutterhead No. 8 socket setting
D009 Cutterhead No. 9 socket setting
D010 Cutterhead No. 10 socket setting
D011 Cutterhead No. 11 socket setting
D012 Cutterhead No. 12 socket setting
D013 Cutterhead No. 13 socket setting
D014 Cutterhead No. 14 socket setting
D015 Cutterhead No. 15 socket setting
D016 Cutterhead No. 16 socket setting
D017 Cutterhead No. 17 pocket setting
D018 Cutterhead No. 18 socket setting
D019 Cutterhead No. 19 socket setting
D020 Cutterhead No. 20 socket setting
D021 Cutterhead No. 21 socket setting
D022 Cutterhead No. 22 socket setting
D023 Cutterhead No. 23 socket setting
D024 Cutterhead No. 24 pocket setting
D025 Cutterhead No. 25 socket setting
D026 Cutterhead No. 26 socket setting
D027 Cutterhead No. 27 socket setting
D028 Cutterhead No. 28 socket setting
D029 Cutterhead No. 29 socket setting
D030 Cutterhead No. 30 socket setting
D031 Cutterhead No. 31 socket setting
D032 Cutterhead No. 32 socket setting
D033 Cutterhead No. 33 socket setting
D034 Cutterhead No. 34 socket setting
D035 Cutterhead No. 35 socket setting
D036 Cutterhead No. 36 socket setting
D037 Cutterhead No. 37 socket setting
D038 Cutterhead No. 38 socket setting
D039 Cutterhead No. 39 socket setting
D040 Cutterhead 40 socket setting
D041 Cutterhead No. 41 socket setting
D042 Cutterhead No. 42 socket setting
D043 Cutterhead No. 43 socket setting
D044 Cutterhead No. 44 pocket setting
D045 Cutterhead No. 45 socket setting
D046 Cutterhead No. 46 socket setting
D047 Cutterhead No. 47 socket setting
D048 Cutterhead No. 48 socket setting

D049 Cutterhead No. 49 socket setting
D050 Cutterhead No. 50 socket setting
D051 Cutterhead No. 51 socket setting
D052 Cutterhead No. 52 socket setting
D053 Cutterhead No. 53 socket setting
D054 Cutterhead No. 54 socket setting
D055 Cutterhead No. 55 socket setting
D056 Cutterhead No. 56 socket setting
D057 Cutterhead No. 57 socket setting
D058 Cutterhead No. 58 socket setting
D059 Cutterhead No. 59 socket setting
D060 Cutterhead No. 60 socket setting
D061 Cutterhead No. 61 socket setting
D062 Cutterhead No. 62 socket setting
D063 Cutterhead No. 63 socket setting
D064 Cutterhead No. 64 socket setting
D065 Cutterhead No. 65 socket setting
D066 Cutterhead No. 66 socket setting
D067 Cutterhead No. 67 socket setting
D068 Cutterhead No. 68 socket setting
D069 Cutterhead 69 socket setting
D070 Cutterhead No. 70 socket setting
D071 Cutterhead No. 71 socket setting
D072 Cutterhead No. 72 socket setting
D073 Cutterhead No. 73 socket setting
D074 Cutterhead No. 74 socket setting
D075 Cutterhead No. 75 socket setting
D076 Cutterhead No. 76 socket setting
D077 Cutterhead No. 77 socket setting
D078 Cutterhead No. 78 socket setting
D079 Cutterhead No. 79 socket setting
D080 Cutterhead No. 80 socket setting
D081 Cutterhead No. 81 socket setting
D082 Cutterhead No. 82 socket setting
D083 Cutterhead No. 83 socket setting
D084 Cutterhead No. 84 socket setting
D085 Cutterhead No. 85 socket setting
D086 Cutterhead No. 86 socket setting
D087 Cutterhead No. 87 socket setting
D088 Cutterhead No. 88 socket setting
D089 Cutterhead No. 89 socket setting
D090 Cutterhead 90 socket setting
D091 Cutterhead No. 91 socket setting
D092 Cutterhead 92 socket setting

D093 Cutterhead 93 socket setting

D094 Cutterhead 94 socket setting

D095 Cutterhead 95 socket setting

D096 Cutterhead 96 socket setting

D097 Cutterhead 97 socket setting

D098 Cutterhead 98 socket setting

D099 Cutterhead 99 socket setting

PLC parameter TMR

T000 spindle shift shutdown time timing

T001 Spindle new gear output to end time delay

T002 Pressure Test Time (ms)

T003 Lubrication Test Time (ms)

T004 Spindle Orientation in Place Completion Delay (ms)

T005 M code execution duration

T006 gear spindle S0~S4 execution time

T007 spindle tool release completes the delay

T008 spindle tool clamping completes the time delay

T009 spindle tool release detection delay

T010 spindle delayed output brake

T011 spindle brake output time

T012 Spindle jog timing

T013 Lubrication opening time (ms) (lubrication is not limited by time when set to 0)

T016 M16 Looser does not detect completion delay ms (default 1000)

T017 M17 clamp knife does not detect completion delay ms (default 1000)

T018 Spindle Directional Signal Delay Detection

T020 spindle tool clamping detection delay

T022 T code execution duration

T023 spindle orientation in place detection delay

T024 M29 switching position mode delay detection

T025 M29 delay completion ms

T027 M28 delay completion ms

T028 M28 switching speed mode delay detection

T030 Manually move the positive direction of the X axis to switch the negative direction delay

T031 Manually move the negative direction of the X axis to switch the positive direction delay

T032 Manually move the Z axis positive direction to switch the negative direction delay

T033 Manually move the Z axis in the negative direction to switch the positive direction delay

T034 Manually move the positive direction of the Y axis to switch the negative direction delay

T035 Manually move the negative direction of the Y axis to switch the positive direction delay

T036 Manually move 4 axes positive direction to switch negative direction delay

T037 Manually move 4 axes negative direction to switch positive direction delay

T038 Manually move 5 axes positive direction to switch negative direction delay

T039 Manually move 5 axes in the negative direction to switch the positive direction delay

T040 indexing table automatic clamping delay output

T041 Indexing table clamping in place delay

T042 Indexing table release into place delay

T043 Indexing table clamped in place to detect delay

T044 Indexing table release into place detection delay

T045 spindle automatic shift detection time

T046 M72 output lead screw clamping delay ms

T047 M72/M73 clamping and loosening complete delay ms

T048 Manually move 6 axes positive direction to switch negative direction delay

T049 Manually move 6 axes negative direction to switch positive direction delay

T050 Pulse Spindle Forward Output Delay Time (ms) (Set 300 when active)

T051 Pulse Spindle Reversal Output Delay Time (ms) (set 300 when active)

T052 Pulse Spindle Stop Off Y4.1 Delay Time (ms) (Set 1000 when active)

T053 Automatic lubrication interval(s)

T100 tool changer manual forward key delay

T101 tool changer manually reverses the key delay

T102 Tool changer rotation stops for time-lapse detection

T103 tool magazine counter delay detection

T104 Tool magazine tool feed/reverse tool delay detection

T105 tool magazine tool retract/return time delay detection

T106 Tool magazine tool feed/reverse tool completion delay

T107 Tool magazine tool retract/return completion delay

T108 tool magazine forward and backward abnormal delay detection

T109 tool changer zero return to place signal in place delay

T110 tool magazine rotation delay detection

T111 tool changer counter in-place delay detection

T112 ATC tool arm in place delay detection

T113 ATC tool arm pull-down rotation into place delay detection

T114 ATC tool arm return to the origin to place delay detection

T115 ATC tool arm output detection delay

The T116 tool changer counter is in place to delay the next step

T117 M22 tool finding detection delay

T118 Position-complete signal disconnect delay

T119 Tool tightness anomaly detection delay

PLC parameter CTR

C004 Panel feed rate count

C005 panel fast magnification counting

C006 Panel spindle magnification counting

C007 panel manual magnification counting

C100 total tool number setting

C110 robot 3-step counter

2022 April 25 5th edition