HNC-8 System Commissioning Manual (Grinding System)

V2.4 Version

Introduction

The manual may help you to quickly get familiar with the HNC-8 system, providing detailed information about commissioning, programming or application methods. Any updates or modification of the manual is not allowed without the written permission of Wuhan Huazhong Numerical Control Co., LTD (hereafter referred to as "HCNC"). Without HCNC's authorization or written permission, any units or individuals are not allowed to modify or correct the manual. HCNC will not be responsible for any losses thus incurred to customers.

In this manual we have tried as much as possible to describe all the various matters concerning of the system. However, we cannot describe all the matters which must not be done, or which cannot be done, because there are so many possibilities. Therefore, matters which are not especially described as possible in this manual should be regarded as “impossible” or “not allowed”.

Please favor me your instruction for shortages and inadequacies of the manual.

Note

- As to notes such as "Limitations" and "Usable functions", the specification provided by the machine tool manufacturer is superior to the manual. Please conduct dryrun before actual machining and confirm machining program, tool compensation volume and workpiece offset, and so on.
- Please explain matters which are not described in the manual as "Infeasible".
- The manual is prepared on the condition that all functions are configured. Please make a confirmation according to the specification provided by the machine tool manufacturer in use.
- For relevant instructions for machine tools, please refer to the specification provided by the machine tool manufacturer.
- Usable screens and functions differ with different NC systems (or versions). Please be sure to confirm specifications before use.

Contents

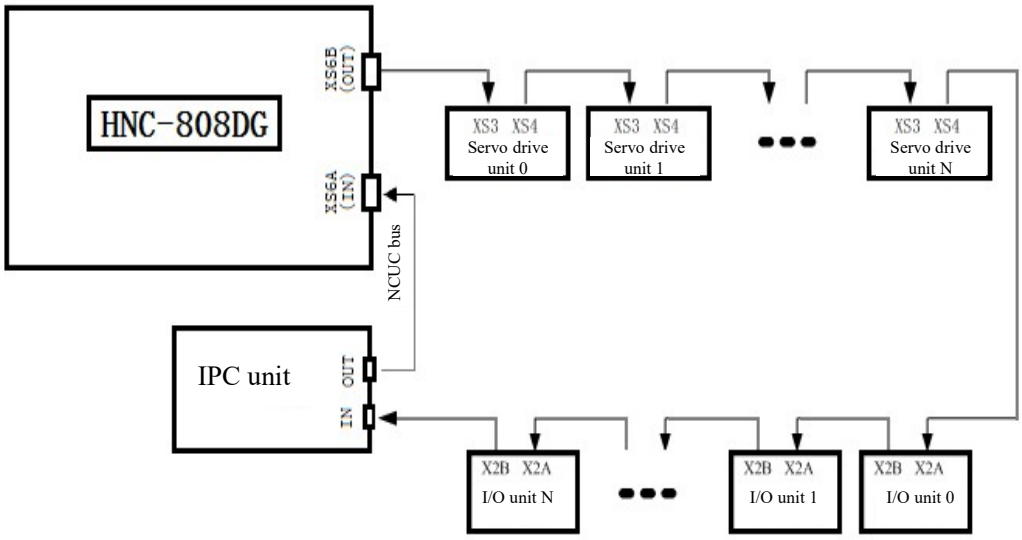
1 HNC-808DG Grinding System Function Introduction

SN	Function	808DG
1	Handwheel simulation	The rotation speed in handwheel replaces the G01 speed. It takes effect when the program runs automatically.
2	Handwheel insertion	Namely "Handwheel interruption", the handwheel interruption amount is superimposed on zero of the currently used coordinate system. It is superimposed on all zero until the interruption amount is cleared.
3	PMC	Programmable Machine Control. Servo axis, but it does not participate in interpolation operation. It has absolute command and relative command. Auxiliary precise control
4	C/S switching	Rotary axis control. Both angle displacement command and speed rotation command can be executed. Selection of grinding head control. It is applicable to thread and screw grinding, crank shaft, cam and gear grinding, etc.
5	Built-in oscilloscope	Built-in speed loop, position loop and notch filter, easy for commissioning.
6	Grating ruler	Support absolute encoder, increment grating. Manufacturer: Heidenhain, Fagor
7	SSTT sampling	Assist sampling commissioning, monitoring and diagnosis
8	★Oblique axis	X axis is the oblique axis and Z axis is regular. Generally, the inclined angle between axes X and Z is 30°. After this function is enabled, Cartesian coordinate system is adopted for programming, namely normal rectangular programming.
9	★Multichannel	808DG has integrated double channels and can be directly used. Used for truss and other schemes of which logical relationships require multi-channel control.
10	PLC online programming	PLC online programming of NC. Also support online debugging of connection of upper and lower microcomputers of PC and NC.
11	★Multigroup D/A	Multi−group D/A, A/D. It is applicable to headstock, frequency conversion wheel, or multigroup frequency conversion output. It can also collect analog voltage and analog current and can be used for monitoring.
12	★Electronic gearbox	Electronic transmission ratio control replaces mechanical transmission ratio. Used for gear machining.
13	★Crankshaft grinding	Tangency point following. The main rotating parts of the engine. After connecting rod is installed, it can change vertical (reciprocating) motion of connecting rod into cyclic (rotational) motion
14	★Cam grinding	Tangency point following. A part in the piston engine. It functions on controlling ON and OFF of air valve
15	★ Video tool setting of grinding wheel	Images collected by HD camera are displayed in the tool compensation interface. Assist observation.
16	★Multiaxis synchronous control	Applicable to complex and high-end grinding.
17	★Thermal error compensation	High precision grinding requirement, thermal error compensation.

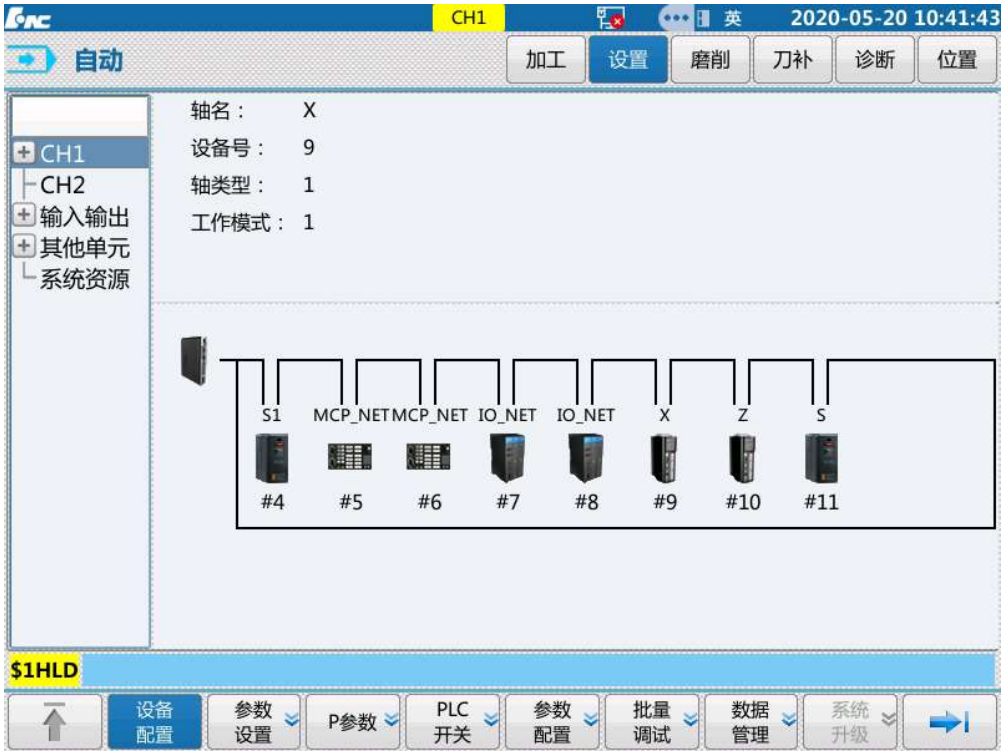
2 Commissioning Steps

2.1 Bus Connection

System, drive and I/O unit are connected in series in proper order.



View devices detected in System Set-Config. Device interface parameters should be set to correspond to the detection sequence one by one.



2.2 Parameter Adjustment and Testing

2.2.1 Whole Structure of Parameter

(1) Assignment of parameter numbers

The number (ID) allocation of various parameters of the HNC-8 CNC system is shown in the following table:

Type	ID	Description
NC parameter	000000 - 009999	Occupy 10000 ID numbers
Machine user parameter	010000 - 019999	Occupy 10000 ID numbers
Channel parameter	040000 - 049999	Divided by channel. Each channel occupy 1000 ID numbers
Coordinate axis parameter	100000 -199999	Divided by axis, each axis occupy 1000 ID numbers
Error compensation parameter	300000 - 399999	Divided by axis, each axis occupy 1000 ID numbers
Device interface parameter	500000 - 599999	Divided by device, each device occupy 1000 ID numbers
Data table parameter	700000 - 799999	Occupy 100000 ID numbers

- NC parameters are the basic parameters of the CNC system, which are used to set parameters of interpolation cycle and operation resolution.
- Machine user parameters are used to set machine tool structure, number of channels and other parameters, such as whether it is a lathe or a milling machine, and the channels used.
- The path through which the channel executes the interpolation movement. Different channels can perform different interpolation motions, and do not affect each other. Dual channel means that two different interpolation motions can be executed at the same time. Channel parameters are used to set the relevant parameters of each channel.
- Coordinate axis parameters are used to set the relevant parameters of the logical axis used in the channel.
- Error compensation parameters are used to set the backlash, pitch error and other related error compensation parameters.
- Device interface parameters are used to set the relevant parameters of physical devices such as axes and I/O.
- Data table parameters are used to set data tables of error compensation, temperature correspondence, etc.

(2) Data type of parameter

The data types of HNC-8 CNC system parameters include the following:

INT4: The parameter value can only be an integer.

BOOL: The parameter value can only be 0 or 1.

REAL: The parameter value can be an integer or a decimal.

STRING: The parameter value is a string of 1 to 7 characters.

HEX4: The parameters are input and displayed in hexadecimal numbers.

ARRAY: The parameters are input and displayed in the form of an array, and each data is separated by "," or ".". The value range of the array elements is 0 to 127.

(3) Parameter access level

- The parameters of each level must be modified and saved after entering the corresponding password and logging in.
- Modification of low-level parameters is allowed after high-level login.
- The curing parameters (access level 5) are not allowed to be modified manually, and are automatically configured by the CNC system (cured at the factory).
- The parameter access levels are shown in the following table:

Access level	Object-oriented	Identity
1	General user	ACCESS_USER
2	Machine manufacturer	ACCESS_MAC
3	CNC controller manufacturer	ACCESS_NC
4	Administrator	ACCESS_RD
5	Curing	ACCESS_VENDER

(4) Parameter activation

There four activation modes of HNC-8 CNC system parameters.

- Save_active: Parameter modification takes effect after SAVE key is pressed.
- Immediate_active: Parameter modification takes effect immediately (mainly used for servo parameter adjustement).
- Reset_active: Parameter modification takes effect after being saved and RESET key is pressed.
- Restart_active: Parameter modification takes effect after being saved and CNC controller is restarted.

2.2.2 NC Parameter

000018	System time display	0: is disabled; 1: is enabled
000020	Automatic display of alarm pop-up window	0: is disabled; 1: is enabled
000023	F feedrate display mode	Selection of display of actual, command and programming speed. When F jitters, view the parameter and change to 1 command or 2 programming
000024	G code line number display mode	0: G code line number is not displayed; 1: G code line number is only displayed on editing interface; 2: G code line number is only displayed on programming running interface; 3: G code line number is displayed both on editing interface and program running interface.
000026	Decimal places of positional value	Recommend to set to 3
000027	Decimal places of speed value	Set based on grinding requirements
000028	Decimal places of rotation speed value	Set based on actual requirements
000030	Screensaver entry waiting minutes	Set based on actual requirements. When 0 is set, the system will never enter screen

000061	Number of T command tool offset compensation number digits	Generally it is 2
000064	Tool wear accumulation Enable	Generally it is 1
000072	Whether machining time display is closed	Display in the bottom right corner of the interface and set as needed

2.2.3 Machine Tool User Parameters

010000	Maximum number of channels	Determine number of channels to be opened in the system
010001	Cutting type of channel 0	Generally it is 1
010002	Cutting type of channel 1	When the second channel is opened, set the parameter
010017	Channel 0 display axis mark [1]	Axes 0-31 correspond to bits 0-31. If 0 is entered in binary, it means there is no axis. If 1 if entered, it means there is an axis. For axes 0, 2 and 5, bits 0, 2 and 5 are set as 1. The value calculated by the system via 2^N is 0X25 in hexadecimal.
010019	Channel 1 display axis mark [1]	When the second channel is opened, set the parameter

010091	Enable #500-#999 being user macro-variables	Only when 1 is set, these 500 variables can be displayed in the macro-variable form and can be saved automatically at the time of power failure
010104	New function debugging	0X100: When this point is valid, for example, assign 2 to #50100, if #50100 is configured as a floating point type, then the value 2 will be converted into a floating point type and assigned to variable #50100; assigned 2.3 to #50100, if #50100 is configured as an integer, then the value 2.3 is converted to an integer 2 and assigned to the variable #50100, and there is a problem of missing floating-point digits. This point is not valid. Integer values can only be assigned to integer variables, and floating-point values can only be assigned to floating-point variables.
010160	F speed display in feed per revolution	1 is set to enable this parameter when the feed per revolution is needed

010170	Corresponding M code of G1007	To call a canned cycle using M code, set corresponding M code
010171-010183 parameters are the same as 010170		
Description: Grinder 808DG opens 32 one-key G registers calling. Correspond to canned cycle in user-defined canned cycle USERDEF.CYC file. Program interval %1006---%1037, the programming format corresponds to G1006-G1037, respectively. Correspondence between G register, program interval and programming G command are as follows: G2637.0-G2637.15%1006---%1021G1006-G1021 G2619.0-G2619.15%1022---%1037 G1022-G1037		

010329	Lubrication time of machine (unit: s)	Set based on actual situation (cooperatively used with "Lubrication" button on panel)
010330	Lubrication stop time (unit: s)	
010331	Three-phase Y-type start time (unit: ms)	5000ms by default
010332	Three-phase Delta-type start time (unit: ms)	6000ms by default, 1s switching interval time
010333	Spindle fluctuation detection time (unit: ms)	Used when there is speed arrival signal
010334	Axis X overload detection time (unit: ms)	Reflected in subprogram S19 of PLC. Set as needed
010335	Axis Z overload detection time (unit: ms)	
010336	PMC axis movement distance (us)	Set in PLC subprogram as needed

2.2.4 Channel Parameter

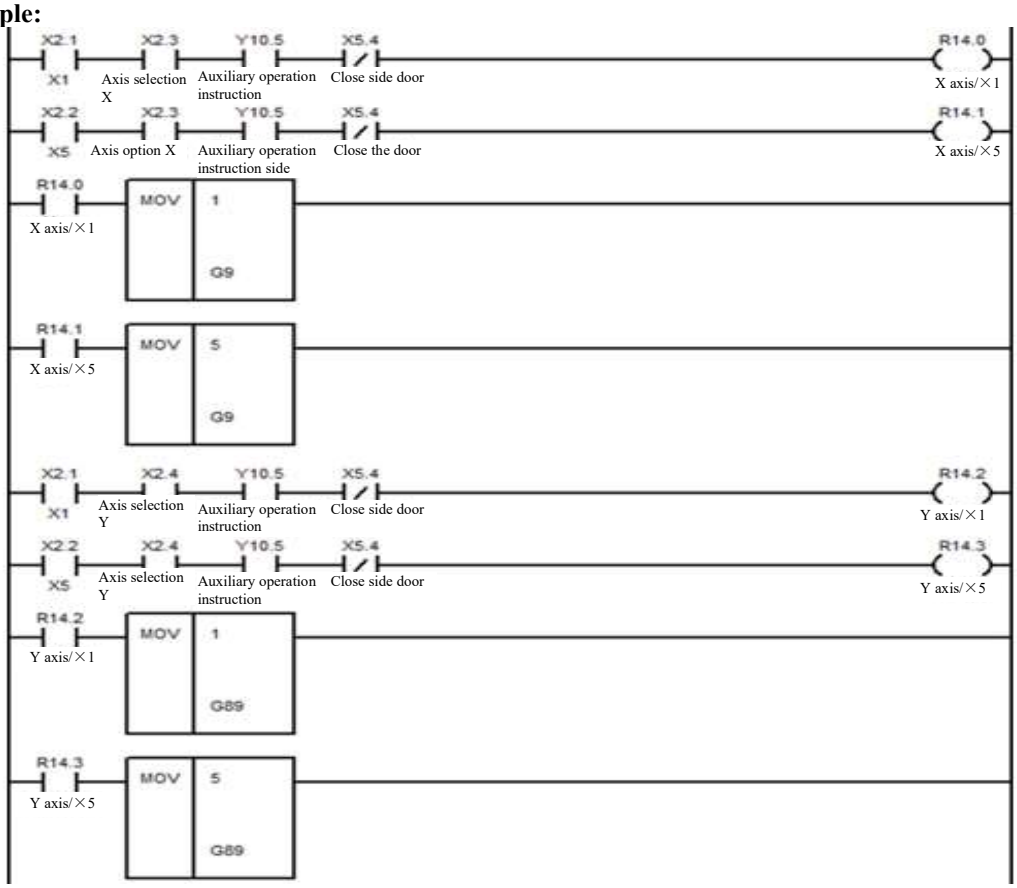
040001-040009	Coordinate axis number	Fill out logical axis number based on actual situation and note the correspondence between that and device interface parameters and the sequence of physical axes in the actual electrical cabinet

040010-040013	Axis number of spindle	Set based on actual need and note the correspondence between the parameter 040028 actual axis number of spindle
040014-040026	Programming name of coordinate axis	Mode name reflected in G code

040027	Display mode of spindle speed	1. When there is an encoder feedback, 0 is set; while configuring 180U asynchronous servo and 160U feed axis as the spindle, set to 0; or when the frequency converter is used, and the encoder is externally connected, set to 0; when no encoder is used, set to 1. 2. Pay attention to correspondence between 040010-040013 and 040028 parameters. e.g.: 040010-040013 correspond to 5, 6, 7 and 8 respectively, and spindles 2 and 3 corresponding to 040012 and 040013 have no encoder feedback. In hexadecimal, it is 1100 (the value C of 040027)
040028	Display axis number of spindle	Set based on 040010-040013. E.g.: 040010-040013 correspond to 5, 6, 7 and 8 respectively, set 040028 to 5, 6, 7 and 8. The sequence is the same as sequence of axes displayed on screen.
040030	Default feedrate of channel (mm/min)	In G code, when the F modal doesn't exist in the previous few lines, it is the default F value of G01.
040032	Diameter programming enable	For the cylindrical grinder, enable this function.
040033	UVW incremental programming Enable	Generally, it is enabled and set as 1, unless the names of U, V and W have been defined as the actual U, V and W axes. E.g. During screw grinding, thread grinding and diamond wheel dressing using a diamond pen, if U and W axes should be used at the same time, at this time U and W are actual axis names and the parameter must be set as 0. Note: G101 and G102 commands should be used for screw grinding. Example: %1234 G54 G90 G1 X100 Z100 U100 W100F1000 G101 X-1 Z-1 U-1 W-1 G102 X1 Z3 U0 W2 G91 G1 X10 Z10 F100 G101 X-1 Z-1 U-1 W-1 G102 X0 Z2 U1 W3 G91 G1 U10 W10 F100 M30

040037-040039	Parameters of handwheel	Set based on actual requirements. If customer needs to modify the magnifications X1, X10 and X100 of handwheel, e.g., modify to X1, X5 and X10, then users change the value of G [axis number *80+9] and assign t G (axis number) *80+9] to 1,5,10 in PLC1.

Example:



040127	Initial tool number	Generally it is set as 1 and the T0101 programming mode is adopted
040128	Number of tools	The parameter affects the number of forms in the tool compensation table

040310	Enable oblique axis control	When there is an oblique axis, the parameter is set as 1.
040311	Quadrature axis number	Z axis by default, parameter value is 2
040312	Oblique axis number	X axis by default, parameter value is 0
040313	Angle of inclination	Generally it is -30. When axis X is vertical to axis Z, taking the position of axis X as the reference, the angle value of the X position is 30 degrees when the actual X axis deviates from the orthogonality, the clockwise direction indicates the negative value, and te counterclockwise direction indicates the positive value.

2.2.5 Coordinate Axis Parameter

(1) Axis Parameter on Control System

Logical axes 0,1--N. According to actual situation, select logical axis number corresponding to axis number set in machine user parameter and channel parameter.

Here descriptions are given with X axis and axis 0 as an example.

100000	Display axis name	The parameter determines axis name displayed in the system interface

100001	Axis type
Description: 0: Not configured, default value. 1: Linear axis. 2: Swing axis, the coordinate value of displayed angle is not limited. 3: Rotary axis, the coordinate value of displayed angle is within the designated range and it will be displayed modulo when the actual coordinates exceeds the set value. 9: When traverse axis is used as spindle, the drive is the feed axis drive. 10: Spindle. Set based on the actually-used drive and the function requirements. It should be specially noted that several parameters in device interface parameters associated with this parameter, as shown in the image below:	

507010	Working mode	0: No position command output 1: Position incremental mode 2: Position absolute mode 3: Speed mode 4: Current mode (torque mode) Generally working mode of feed axis is set as 1 and that of spindle is set as 3
507014	Feedback position loop enable	0: Feedback position does not adopt cycle counting mode 1: Feedback position adopts cycle counting mode 2: This mode is adopted when feed axis servo is switched to spindle The parameter should be set as 0 for linear feed axis and swing axis and 1 for rotary axis and spindle

100004	Numerator of electronic gear ratio [displacement] (um)	① Linear axis: Motor connects directly to screw rod. 1:1 transmission ratio, 100004 is the thread pitch of screw rod

100005	Denominator of electronic gear ratio [pulse]	in the unit mm; 100005 is the number of pulses per revolution of motor and common type of Huada motor is 131072. If transmission ratio between motor and screw rod is not 1:1, it is necessary to multiply the actual mechanical transmission ratio on the basis of the numerical ratio in the case of the above direct connection. ② Speed axis: When mechanical transmission ratio is 1:1, 100004 value is 360000 (s), 100005 is the number of pulses per revolution of motor and common type of HCNC asynchronous servomotor is 4096. If transmission ratio is not 1:1, it is necessary to multiply the actual mechanical transmission ratio. ③ Switching feed axis to spindle: Transmission ratio is calculated using the same method in ②. ④ Swing axis, rotary axis: Transmission ratio is calculated using the same method in ②.
100006	Positive software limit coordinate	For software limit, it should be noted that after "Enable diameter programming", software limit of diameter axis is 1/2 of the displayed coordinate value
100007	Negative software limit coordinate	

10010-100016	Parameters of reference point return	Set based on actual situation; it should be noted that for 100012 encoder feedback offset amount, enter the corresponding axis number in "Auto offset" under "Set" module. Press the emergency stop button and reset to validate the setting. As shown below, set zero point of axis.

Axis zero setting

10032-100039	Axis speed, acceleration/deceleration adjustment	Calculate according to actual thread pitch, transmission ratio and maximum motor speed. It should be stressed that while switching spindle to feed axis, the speed should be set as the maximum allowable parameter value; otherwise, rotation speed S cannot increase.

10043-100047	Parameters of handwheel	Refer to the relevant explanations in the channel parameters

100050	Default S speed (rad/min)	Default manual speed, set based on actual need
100055	Feed spindle orientation angle (degree)	Parameters of orientation and zero speed when the feed axis is used as the spindle. Set based on field commissioning situation.
100056	Tolerance of feed spindle zero speed (pulse)	

100060	Positioning tolerance (mm)	Set based on actual situation. When the value is 0, the system does not detect positioning tolerance.
100061	Maximum tracking error of 1m/min (mm)	Fill out according to actual situation. When the value is 0, the system does not detect following tolerance.
100062	Pulse count per revolution of axis (pulse)	Pulse count per revolution of motor when mechanical transmission ratio is 1:1. The parameter is often set to the same value as the parameter Number of Feedback Position Loop Pulses in the device interface parameter. If numerator and denominator of electronic gear ratio in the axis parameters have no reduction of a fraction, it also has the same value as "Denominator of Electronic Gear Ratio" in the axis parameters.
100068	Screw rod lead (mm)	Set based on actual thread pitch, take effect while shifting gear stages
100073	Displayed coefficient of	When the axis type is rotary axis, modify the

	rotary axis speed	parameter. Conversion between degree/min and rev/min

(2) Axis Parameter on Servo

Common commissioning parameters

P0	Position proportional gain	Set based on actual situation
P2	Speed proportional gain	
P4	Speed feedback filter coefficient	
P24	Number of motor pole pairs	
P25	Rotary axis speed display coefficient	
P26	Encoder zero offset	
P27	Current proportional gain	
P43	Specification of 160/180UD drive. For 180US drive, corresponding parameter is P59	

2.2.6 Device Interface Parameter

Analog spindle of device 4		
504010	Working mode	Generally it is set as 3, speed working mode.
504011	Logical axis number	Refer to the axis number corresponding to spindle set in the channel parameters.
504012	Inverse encoder feedback mark	If the mark of feedback S is not the same as the actual situation, modify the parameter
504013	Spindle DA output type	Select 0-10V or -10V-10V based on actual situation
504014	Spindle DA output zero drift adjustment amount (mv)	The measured DA output value when the spindle is at M3S0, in the nit mv
504015	Pulse count of feedback position loop	Generally it is 4096, and it should be set according to actual situation
504016	Spindle encoder feedback device number	Set based on the device number of control panel feedback interface
504017	Spindle DA output device number	Device number occupied by electrical IO
504018	Spindle encoder feedback interface number	Calculate the number of shifted groups (16 bits as a group) according to 504016 and electrical IN signal.
504019	Axis DA output port number	Calculate the number of shifted groups (16 bits as a group) according to 504017 and electrical OUT signal.

MCP unit of 808DG system occupies 2 devices: device 5 and device 6

Device 5

505010	MCP type	7
505012	Initial group number of input point	480
505013	Number of input point groups	10
505014	Initial group number of output point	480
505015	Number of output point groups	10
505016	Inverse handwheel direction mark	If the moving direction of the handwheel axis is opposite to the actual direction, modify this parameter
505018	Band switch encoding type	Band switch type selection

Device 6

506010	MCP type	7
506012	Initial group number of input point	490
506013	Number of input point groups	10
506014	Initial group number of output point	490
506015	Number of output point groups	10
506019	Number of additional analog spindles	When more than one analog spindles are needed, modify this parameter. If two groups of analog spindles are needed, number of additional analog spindles is 1

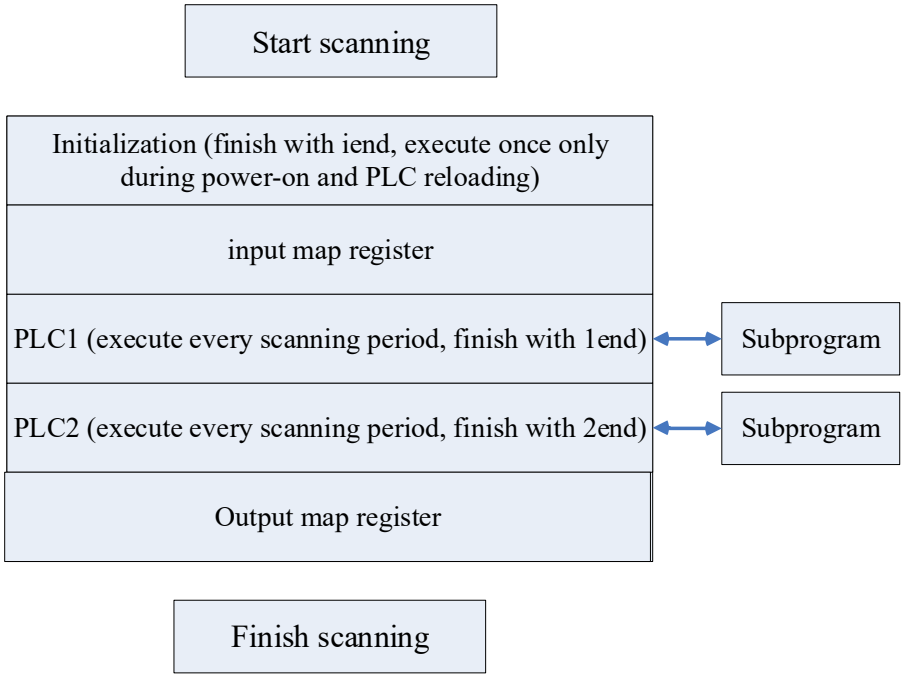
Device parameters corresponding to axes should be set according to axis configuration in channel parameters.

507010	Working mode	Set based on actual situation
507011	Logical axis number	Set based on actual situation
507012	Inverse encoder feedback mark	When the displayed rotate speed of the spindle is opposite to the actual rotation direction, the parameter can be set as 1
507014	Feedback position loop mode	Pay attention to the value when the feed axis is switched to the spindle
507015	Pulse count of feedback position loop	Generally it is the pulse count of axis per revolution, note whether mechanical transmission ratio is 1:1 and refer to relevant 2 parameters in axis parameters
507016	Encoder type	Generally the absolute value is 3

2.3 PLC Commissioning

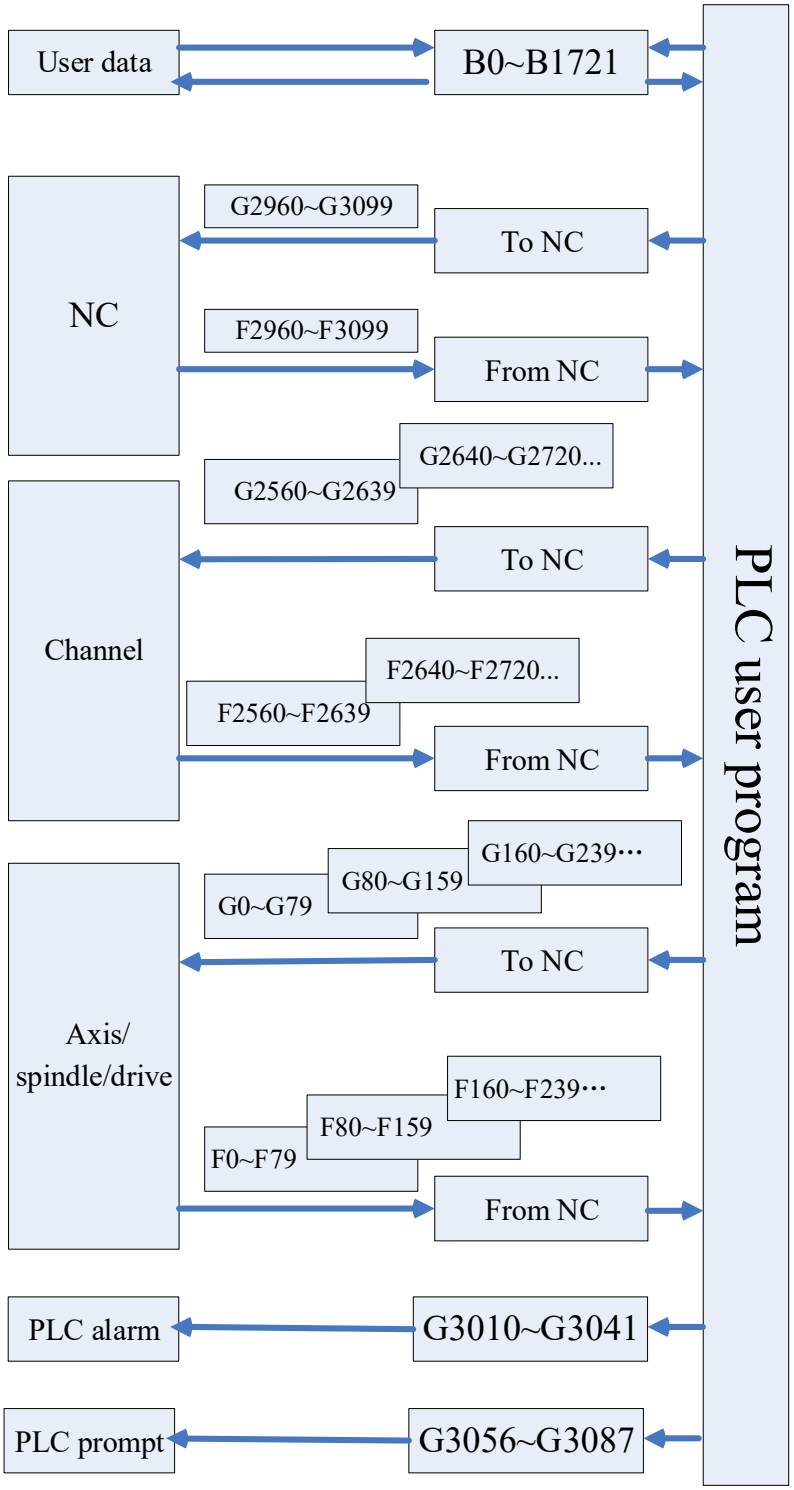
2.3.1 HNC-8 PLC Structure

HNC-8 ladder diagram PLC adopts cyclic scanning mode. In the beginning of program execution, PLC will be initialized once when it is powered on or reloaded. Then all inputted states are transmitted to the input map register. Afterwards, call user programs PLC1 and PLC2 in sequence. When a scanning cycle is completed, all results will be transmitted to the output map register to control actual output of PLC, and so on.



2.3.2 Working Principle of PLC Interface Signal

PLC interface signal is responsible for information exchange between PLC and NC.



- F register is a state flag register used to input CNC input signal to PLC control module from CNC.
- G register is a control flag register used to output CNC output signal to CNC from PLC control module and the signals are processed by CNC.
- B register is a data register (saving data after power off), and the value of the register still remains at the state before power-off after power-off. The data register can also be used as PLC parameter and users can define usage of each parameter.

2.3.3 PLC Specification

Specification	HNC8
Programming language	Ladder, STL
The first-level program execution cycle	1ms
Program capacity Ladder diagram Statement list Symbol name	 5000 lines 10000 lines 1000 pcs.
Command Basic command, function command	
Single-byte internal relay (R) Dual-byte internal register (W) Four-byte internal register (D) Timer (T) Counter (C) Subprogram (S) Mark number (L) User-defined parameter (P) Single-byte internal register (I) Single-byte internal register (Q) Holding-type storage area Four-byte register (B) Holding relay (K)	2048 bytes (R0-R2047) 512 bytes (W0-W255) 1024 bytes (D0-D255) 512 (T0-T511) 512 (C0-C511) 253 (S0-S252) 10000(L0-L9999) 700(P0-P699) 128 bytes (I0-I127) 128 bytes (Q0-Q127) 6888 bytes (B0-B1721) 128 bytes (K0-K15)
I/O module (X) (Y)	X0-X511 Y0-Y511

2.3.4 PLC Switch File and P Parameter

(1) PLC switch

Set → PLC switch

This part is all mapped to PLC ladder diagram and should be opened according to actual needs.

Correspond to P196 point. Parameters 1-32 correspond to P196.0-P196.31. For example, parameter 10is the P196.9. This part is used for auxiliary debugging including shielding and emergency stop, etc.

(2) P parameter

Note: Corresponding P parameters can also be modified here.

2.3.5 Common PLC Module

(1) G31 module

Specify the axis moving after G31 command just like specifying linear interpolation in G01. If an external skip signal is inputted when this command is executed, the command execution is interrupted and the next program block is executed.

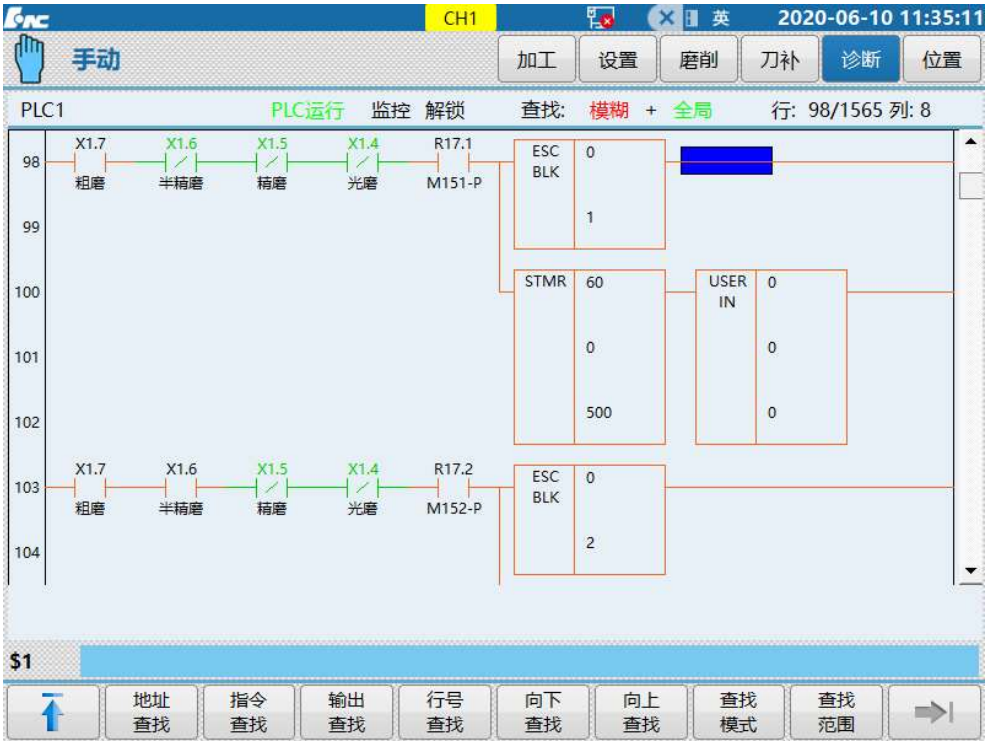
Format

G31 L_IP_; Figure after L is the contact point number, which is consistent with the contact number in PLC

G31: Non-modal G code

(2) USERIN

As shown above, user-defined input. Used to judge whether signal arrives, that is, the value of #1190 is reflected by G code.



(3) VARGET and VARSET modules

- **Variable type**

Variable type <Address 1>	Variable offset number <Address 2>
0: User variable	0 to 4999 correspond to #50000 to #54999
1: Extended user variable	0 to 499 correspond to #500 to #999
2: 32-bit integer system variable	0 to 9999 correspond #40000 to #49999
3: 64-bit integer system variable	0 to 4999 correspond to #40000, #40002…#49998
4: Floating point system variable	0 to 4999 correspond to #40000, #40002…#49998
5: 32-bit integer channel variable	0 to 7999 correspond to #0 to #7999
6: Floating point channel variable	0 to 3999 correspond to #0, #2…#7998
7: 32-bit integer axis variable	0 to 9999 correspond to #60000 to #69999
8: 64-bit integer axis variable	0 to 4999 correspond to #60000, #60002…#69998
9: Tool variable	0 to 19999 correspond to #70000 to #99999

Note:

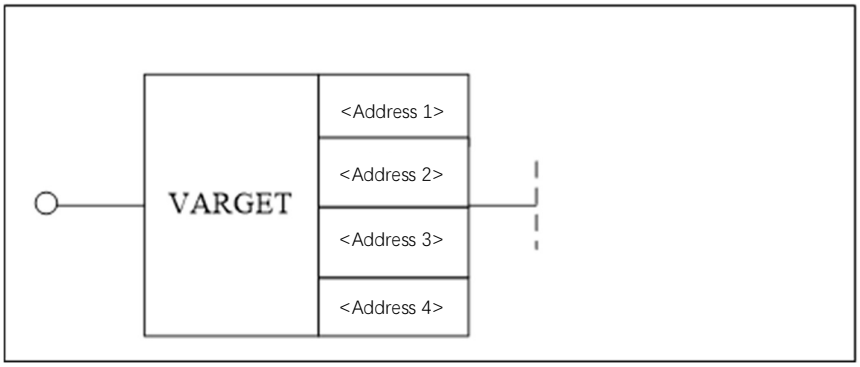
① Channel variables, each channel occupies 2000 variables. That is, #0 to #1999 are channel 0 variables, #2000 to #3999 are channel 1 variables, #4000 to #5999 are channel 2 variables, and #6000 to #7999 are channel 3 variables.

② Axis variables, each axis occupies 100 variables, that is, #60000to #60100 are axis 0 variables, and so on.

③ Tool variables, each tool occupies 200 variables, that is, #70000 to #70200 are the variables of the No. 1 tool, and so on.

- **VARGET (read system variable value)**

Format



Parameter	Storage area	Description
<Address 1>	Constant	Variable type, ranges from 0 to 9.
<Address 2>	Constant, X, Y, F, G, R, W, D, P, B	The offset of the variable. The value increased of the starting variable number corresponding to the variable type selected in <Address 1>.
<Address 3>	Constant	The exponent of 10. It is used to enlarge the value of the variable.
<Address 4>	Y, G, R, W, D, B	The register address of the value that needs to be assigned to the system variable.

①Function description

The PLC reads various variable values of the system and assigns them to registers.

②Parameter description

Parameter 1: Variable value type;

Parameter 2: The offset number of the read variable address (it can be a constant or the value in a register);

Parameter 3: Floating-point variable increases the multiples of 10 (that is, 10 to the power of N, N is the value of the parameter);

Parameter 4: The address where the value of the variable to be read is stored

③Usage instructions

The number of the macro variable being read = the starting variable number corresponding to <address 1> + the offset number of <address 2>.

The assigned register is the register filled in <Address 4>.

The assigned value is the value of the macro variable*10 to the power of N (N is the value in <address 3>)

Example

Increase the value of floating-point user macro variable #51242 by 100 times (10 to the power of 2) and assign it to register D30.

The first parameter: 0 represents the data type. This place indicates that the variable in the interval #50000-#54999 is selected.

The second parameter: 1242 represents the #51242 variable

The third parameter: 2 indicates square of 10, which is 100.

The fourth parameter: D30 means: the value in #51242 is taken out, and multiplied by 100 (the third parameter), and then stored in the register D30.

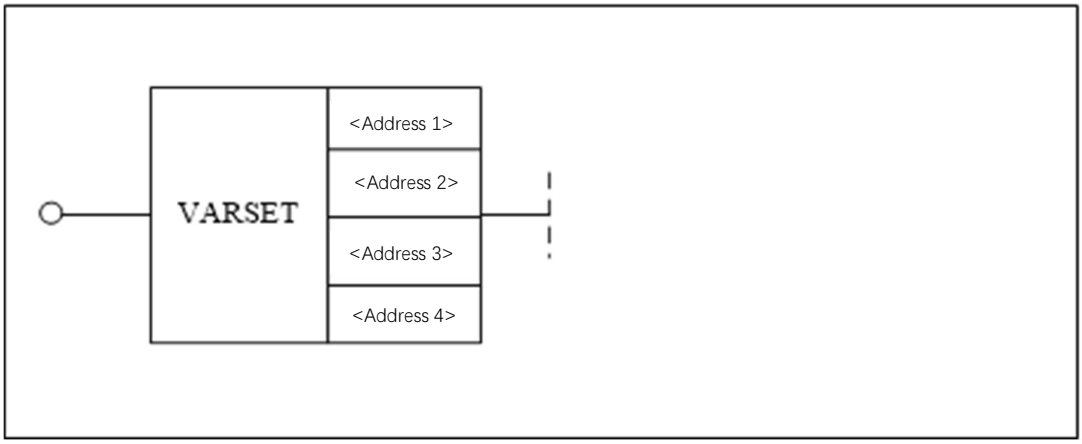
Note: ①If the data type of #51242 is INT, and 0,1,2.... is filled in for the third parameter according to actual needs, that means, 1 times, 10 times, 100 times, and 10 to the Nth power.

②If the data type of #51242 is FLOAT, and it is a four-digit decimal, then the third parameter is 4 or a value greater than 4 according to actual needs, that is 10000 times. The value of #51242 is multiplied by 10000, which si converted from a decimal to an integer, and stored in the D register.

③ HCNC control system PLC cannot directly process numerical values with decimals, but can only process integer values. Therefore, as mentioned in item ②, if #51242 is a decimal number, it must be converted into an integer value to participate in the PLC calculation.

- VARSET (set various of variable values of system)

Format



Parameter	Storage area	Description
<Address 1>	Constant	Variable type, cam be 0, 1, 2, 9
< Address 2>	Constant, X, Y, F, G, R, W, D, P, B	The offset of the variable. The value increased of the starting variable number corresponding to the variable type selected in <Address 1>.
< Address 3>	Constant	The exponent of 10. It is used to reduce the value of the variable.
< Address 4>	Y, G, R, W, D, B	The register address of the value that needs to be assigned to the system variable.

①Function description

Set the variable value of the system through the register in the PLC.

②Parameter description

Parameter 1: Variable value type;

Parameter 2: The offset number of the variable address that needs to be set (it can be a constant or a value in a register);

Parameter 3: The value is reduced by the multiples of 10 (that is, the Nth power of 10, N is the value of this parameter);

Parameter 4: The register address of the value that needs to be assigned to the system variable

③Usage instructions

The assigned macro variable number = the starting variable number corresponding to <address 1> + the offset number of <address 2>.

The register used for assignment is the register filled in <address 4>.

The assigned value = the value of the macro variable / 10 to the power of N (N is the value in <address 3>)

Example

The first parameter: 0 represents the data type. This place indicates that the variable in the interval #50000-#54999 is selected.

The second parameter: 1425 represents #50000 offset 1425, which is the variable #51425

The third parameter: 0 represents 10 to the 0 power, which is 1.

The fourth parameter: Divide the value of D31 by the Nth power of 10 (N is the value of the third parameter), and the result is assigned to the macro variable #51425 for the numerical calculation of the macro variable in the G code.

Note: ①The third parameter is the value 0 for the current example.

Divide the value of D31 by the Nth power of 10 (N is the value of the third parameter), and the result is assigned to the macro variable. If #51425 in the G code is of type INT, use it directly in the G code; if #51425 in the G code is FLOAT type, use it directly in the G code, and divide by the corresponding value according to the actual situation, such as 10,100,1000,1000, and then perform the calculation with it.

2.3.6 PLC Point

Input point:

X0.0	Wheel alarm	X2.0	Radial 1 measuring instrument advances to the right position
X0.1	Oil pump alarm	X2.1	Radial 1 measuring instrument retracts to the right position

X0.2	Water pump alarm	X2.2	Radial 2 measuring instrument advances to the right position
X0.3		X2.3	Radial 2 measuring instrument retracts to the right position
X0.4	PWM spindle alarm	X2.4	The end face measuring instrument advances to the right position
X0.5		X2.5	The end face measuring instrument retracts to the right position
X0.6	Air conditioner alarm	X2.6	Air switch
X0.7		X2.7	External emergency stop
X1.0	Radial measuring instrument P1	X3.0	
X1.1	Radial measuring instrument P2	X3.1	
X1.2	Radial measuring instrument P3	X3.2	
X1.3	Radial measuring instrument P4	X3.3	The tailstock advances to the right position
X1.4	End face measuring instrument	X3.4	The tailstock retracts to the right position
X1.5		X3.5	Zero speed signal
X1.6		X3.6	Speed arrival signal
X1.7		X3.7	

Output point:

Y0.0	Headstock rotation	Y2.0	
Y0.1	Wheel rotation	Y2.1	
Y0.2	Three-phase Y type start	Y2.2	
Y0.3	Three-phase Delta type start	Y2.3	
Y0.4	Cooling	Y2.4	
Y0.5	Oil pump	Y2.5	
Y0.6	Lubrication	Y2.6	
Y0.7	Work light	Y2.7	
Y1.0	Outer diameter measuring instrument stretching and retract	Y3.0	

Y1.1	End face measuring instrument stretching and retract	Y3.1	
Y1.2		Y3.2	
Y1.3		Y3.3	
Y1.4		Y3.4	
Y1.5		Y3.5	
Y1.6		Y3.6	
Y1.7		Y3.7	

2.3.7 M Code

M code	Desciption	M code	Desciption	M code	Desciption
M3	Headstock ON	M25	Servo sampling ON	M52	Thread OFF
M5	Headstock OFF	M26	Servo sampling OFF	M90	User input
M7	Cooling ON	M30	Program end	M91	User output
M9	Cooling OFF	M33	Wheel ON	M92	Manual intervention ON
M12	Radial 1 measuring instrument 1 extends	M35	Wheel OFF	M93	Manual intervention OFF
M13	Radial 1 measuring instrument retracts	M44	Chuck clamping	M150	Cancel measuring instrument signal
M14	Radial 2 measuring instrument extends	M45	Chuck release	M151	Radial 1 measuring instrument P1
M15	Radial 2 measuring instrument retracts	M46	Oil pump ON	M152	Radial 1 measuring instrument P2
M16	End face measuring instrument stretching	M47	Oil pump OFF	M153	Radial 1 measuring instrument P3
M17	End face measuring instrument retract	M48	Tailstock stretching	M154	Radial 1 measuring instrument P4
M19	Orientation ON	M49	Tailstock retract	M161	End face measuring instrument P1
M20	Orientation OFF	M51	Thread ON	M162	End face measuring instrument P2

2.4 Common G Code Command

2.4.1 G10 Command

G90(G91)G10 L2PpIP

Parameter	Meaning
Pp	Specify workpiece origin offset of relative workpiece coordinate systems 1-6: ➢ 1 corresponds to G54 workpiece coordinate system ➢ 2 corresponds to G55 workpiece coordinate system ➢ 3 corresponds to G56 workpiece coordinate system ➢ 4 corresponds to G57 workpiece coordinate system ➢ 5 corresponds to G58 workpiece coordinate system ➢ 6 corresponds to G59 workpiece coordinate system
IP	For absolute programming, it is the workpiece origin offset of each axis For incremental programming, accumulated to the original workpiece origin offset of each axis

G90(G91)G10 L14 Pp X_ Z_ ; format of cumulative wear value: 10L14U_W_

Parameter	Meaning
Pp	Tool offset number
X	Tool compensation data X
Z	Tool compensation data Z

2.4.2 Stop Read-ahead

(1) G08

If the command is encountered during program execution, the system stops the interpretation of the subsequent lines and continues to interpret and run only after the previous interpreted commands are executed. The command is frequently used in real-time coordinate reading and status determination.

(2) G31 L-2 Kx。

Stop read-ahead of corresponding axes. The value of X after K is 2^N and N is the axis number. It is applicable to reading coordinate variables after skip.

2.4.3 Other Codes

G108, G109 C axis mode switching

G101, G102 Release and acquisition of axis

G110 Custom alarm usage

M98, G65 When users write the macro program by themselves, they can choose to use it according to the actual situation

2.5 New Functions in Version 2.40.01

2.5.1 Technology Encryption Function

After the canned cycle file TURNING.CYC and the custom cycle file USERDEF.CYC are encrypted, the TURNING.CYC and USERDEF.CYC programs on the system are invisible and cannot be edited. After TURNING.CYC and USERDEF.CYC files are opened, they show garbled characters, and the technology program can only be seen after decryption.

Canned cycle files TUINING.CYC and USERDEF.CYC cannot be loaded and exported through data management.

The program written by users can also be encrypted to protect the users' technology.

After encryption, the program is invisible and cannot be edited.

Encryption program is running

2.5.2 System Display Manual

As shown below in the figure, press Grind → User file

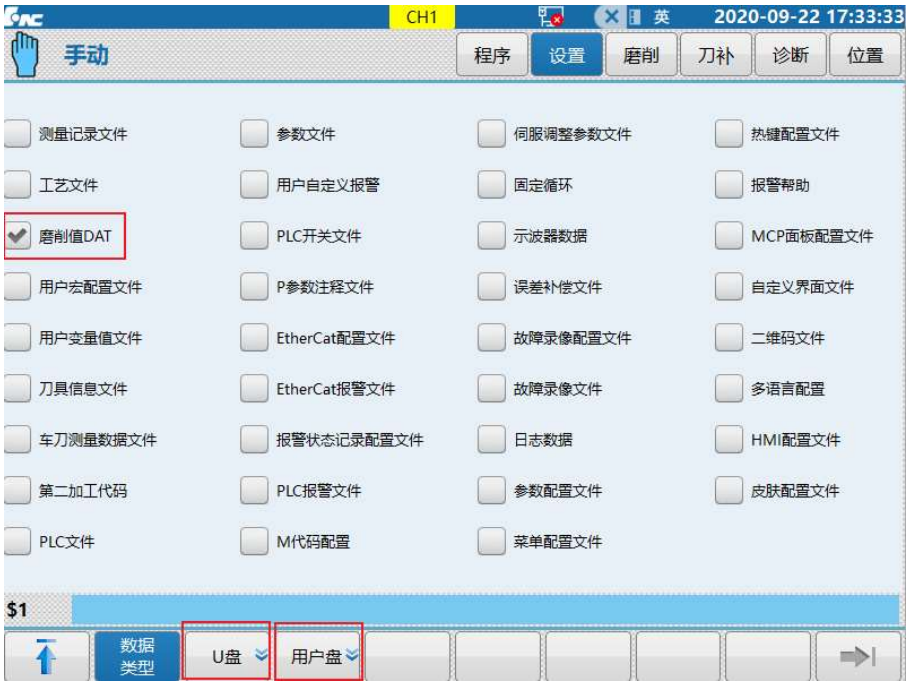
Click the submenu Open File to open corresponding manual to view the manual. The up and down direction keys can be used to turn pages and locate pages.

The WORD manual written by users can be upgraded to the system, and the manual can be viewed on the system.

2.5.3 Backup and Restore of Grinding Data

Press Set → Data manage to select Grinding Value DAT to back up and restore workpiece grinding data. The data can be backed up to U Disk or User Disk. Users do not need to fill in the grinding process variable value again when replacing the workpiece, but only needs to import and export the grinding process file, which is

convenient for the user to replace the workpiece

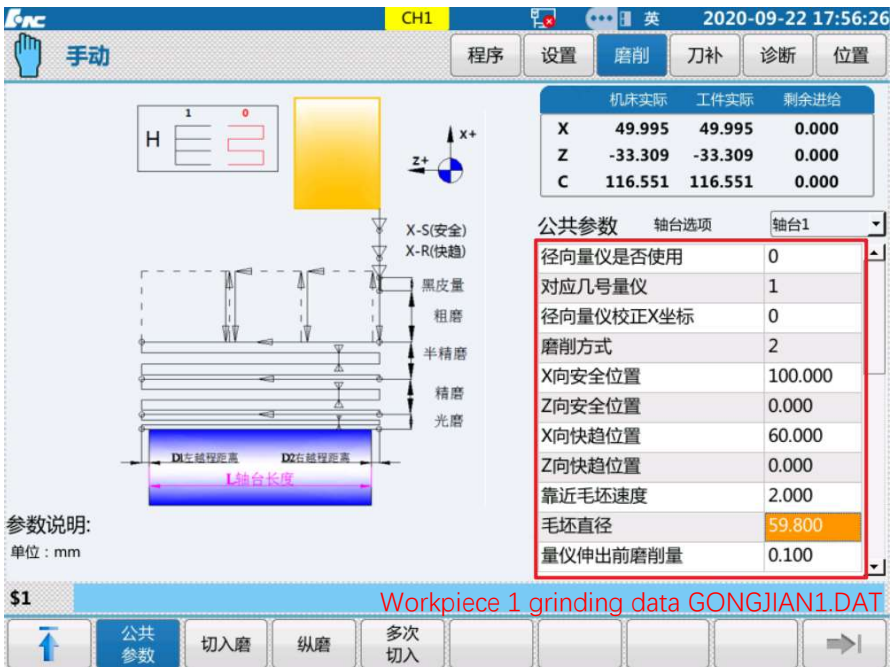


The menu Technology Package menu is for the technology data backup, and the menu Process Restore is for the technology data restoration.

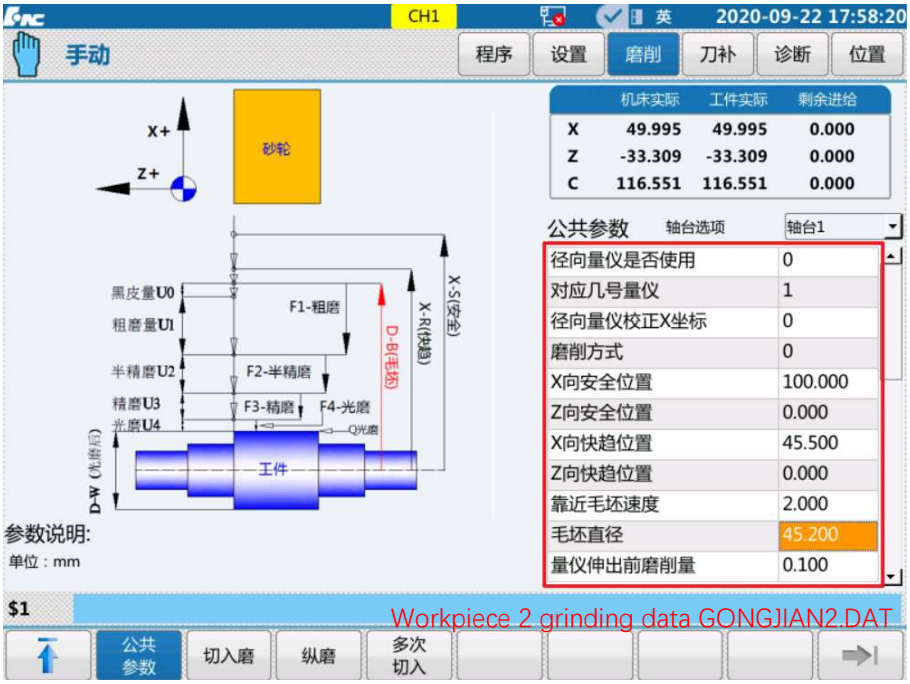
E.g:

For workpiece 1, the technology grinding data is backed up as GONGJIAN1.DAT file through the Technology Package function.

For workpiece 2, the technology grinding data is backed up as GONGJIAN2.DAT file through the Technology Package function.


CH1
英
2020-09-22 17:56:26
手动
程序
设置
磨削
刀补
诊断
位置
机床实际
工件实际
剩余进给
X 49.995 49.995 0.000
Z -33.309 -33.309 0.000
C 116.551 116.551 0.000
X-S(安全)
X-R(快趋)
黑皮量
粗磨
半精磨
精磨
光磨
D1左越程距离
D2右越程距离
L轴台长度
公共参数
轴台选项
轴台1
径向量仪是否使用 0
对应几号量仪 1
径向量仪校正X坐标 0
磨削方式 2
X向安全位置 100.000
Z向安全位置 0.000
X向快趋位置 60.000
Z向快趋位置 0.000
靠近毛坯速度 2.000
毛坯直径 59.800
量仪伸出前磨削量 0.100
参数说明:
单位：mm
$1
Workpiece 1 grinding data GONGJIAN1.DAT
公共参数
切入磨
纵磨
多次切入


CH1
英
2020-09-22 17:57:19
手动
程序
设置
磨削
刀补
诊断
位置
系统盘 磨削值DAT ../data
名称 大小 修改时间
..
GRINDDATA 2020-08-31 03:11:20
grinddata 2020-08-28 05:50:01
332.DAT 87KB 2020-08-20 15:59:14
10.DAT 87KB 2020-08-20 15:57:53
U盘 磨削值DAT /g
名称 大小 修改时间
..
0g2637 2020-09-11 15:15:44
0820mi 2020-08-20 14:17:58
zairushuomingshu 2020-08-19 11:08:06
1.0jiami 2020-07-20 10:16:36
$1
磨削工艺所有变量GONGJIAN1.DAT备份，成功!
载入
备份
删除
重命名
工艺打包
工艺还原
窗口切换

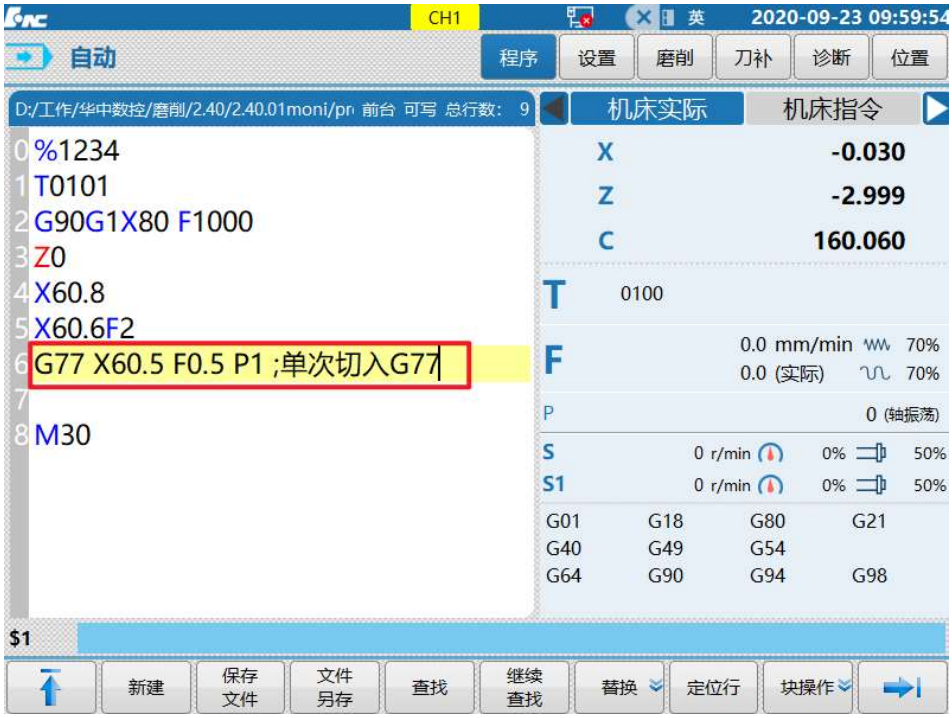


Import the technology grinding data of corresponding workpiece through Technology Restore

2.5.4 Insert Cycle

Press PROG → Edit to insert cycle.

The Insert Cycle function includes common grinding canned cycles. There is a corresponding schematic diagram on the right side of each parameter to facilitate users to understand and fill in the grinding parameter values. After filling in the parameters, press NC Output to output the canned cycle program to the program editing cursor. The Insert Cycle function is convenient for users to write canned cycle programs, users do not need to memorize the meaning of each parameter, just fill in the parameters according to the schematic diagram.



2.5.5 Grinding Card Interface Refreshing

Different technology parameters correspond to different schematic diagrams, which is convenient for users to understand and fill in the technology parameters.

2.5.6 Oblique Axis

(1) During tool setting, the workpiece coordinates are updated in real time

(2) Tool setting can be performed to oblique axis like straight axis. Users can enter pre-grinding diameter for X-direction tool setting, then retract on X axis, and enter pre-grinding length for Z-direction tool setting.

(3) Oblique axis record coordinate tool setting function

Calculate the X-axis and Z-axis offset according to the recorded coordinates.

2.5.7 G153/G154 Axis Oscillation Function

Command of turning on axis oscillation: G153 IP_ P_

Command of turning off axis oscillation: G154 IP0

Note: IP is the axis that needs to turn on the oscillation function, such as X, Z; IP is followed by the oscillation distance, positive and negative directions; P_ is the oscillation axis speed.

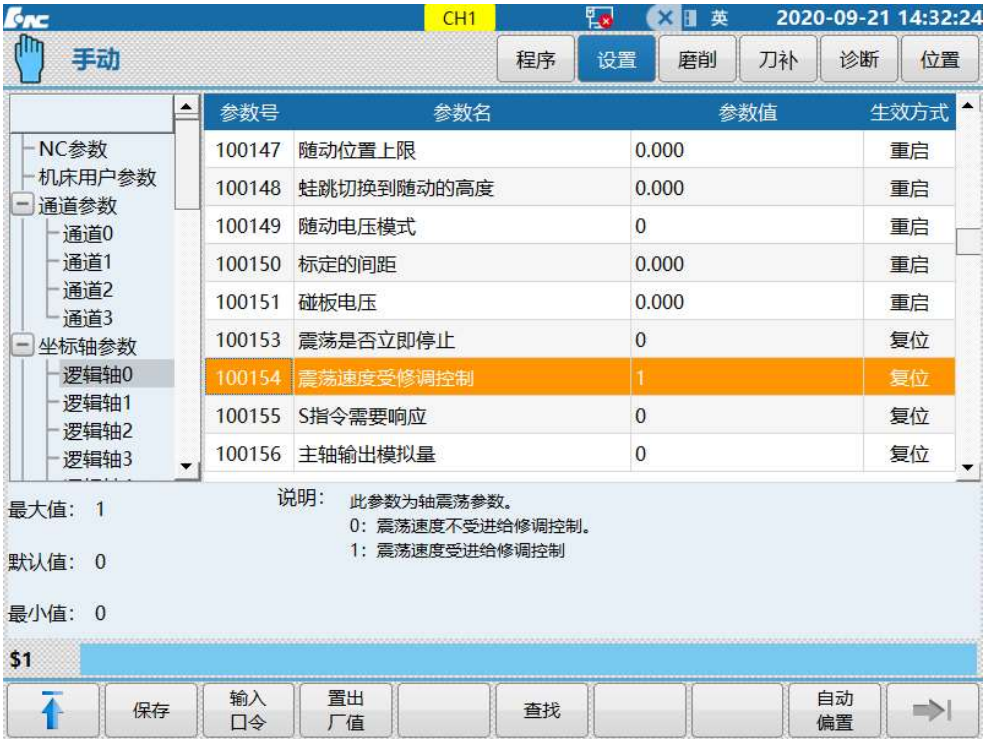
For example,

```
%123
T0101
G90 G1 Z0 F1000
G153 Z50 P2000    ; Z-axis oscillation is turned on, Z axis oscillates between 0 and 50.
G4X5
G154 Z0           ; Oscillation is turned off.
M30
```

System parameters related to axis oscillation:

10X153 Whether stop axis oscillation immediately

10X154 Oscillation speed is controlled by override



2.5.8 Workpiece Coordinate System Zero Translation Commands G155, G156

Format: G155 U_ W_ ; Workpiece coordinate system zero translation

G156; Cancel translation

Or G155 U_ W_ P_ ; Workpiece coordinate system zero translation of specified tool number

G156 P_; Cancel translation of specified tool

Description: U_W_ is the translation increment corresponding to X axis and Z axis. P_ is the tool number, for example, P1 is T0101, P2 is T0102.

G155 U_W_ can only perform the incremental translation of current zero of the workpiece coordinate system established by T command, and the translation value only takes effect when encountering T command.

G156 cancels the translation of all workpiece coordinate systems.

G155 U_W_ P_ can only perform the incremental translation of current zero of the workpiece coordinate system established by T command for specified tool, and the translation value only takes effect when encountering T command.

G156 P_ cancels the translation of the workpiece coordinate system established by the T command for the specified tool number.

For example,

```
%1
G156 P1; Cancel the translation of tool 1
T0103; Tool setting with end face measuring instrument
G01 X50 F2000
G01 Z-10
M151
G31 G01 L1 W-10 F200
W2
G31 G01 L1 W-5 F10
#54005=#1022-#1032; Take the error value of each detection by the end face measuring instrument
G155 P1 W[#54005];Translate the error value detected by the end measuring instrument each time to the grinding tool T0101
G01 Z-10
T0101; Grinding tool
G01 X50 F2000
Z0
.....
M30
```

3 Grinding Technology Introduction

3.1 Canned Cycle

3.1.1 Plunge Grinding

Single-times plunge grinding: G77 X（U）_ F_ P_ A_ O_

X (U) — X is the target coordinate value. G90 is the absolute programming mode, and G91 is the incremental programming mode. U is a value relative to start point of grinding, and ± represents directions. (Diameter programming is by default. If it needs to be modified, modify diameter/radius mode in system parameters)

F — Feedrate of grinding wheel (X direction).

P — Measuring instrument signal. If P value is set, it means that the measuring instrument is used; otherwise, it means that measuring instrument is not used. P1 represents the rough grinding in-position signal. P2 represents the semi-fine grinding in-position signal. P3 represents the fine grinding in-position signal. P4 represents the dimension signal.

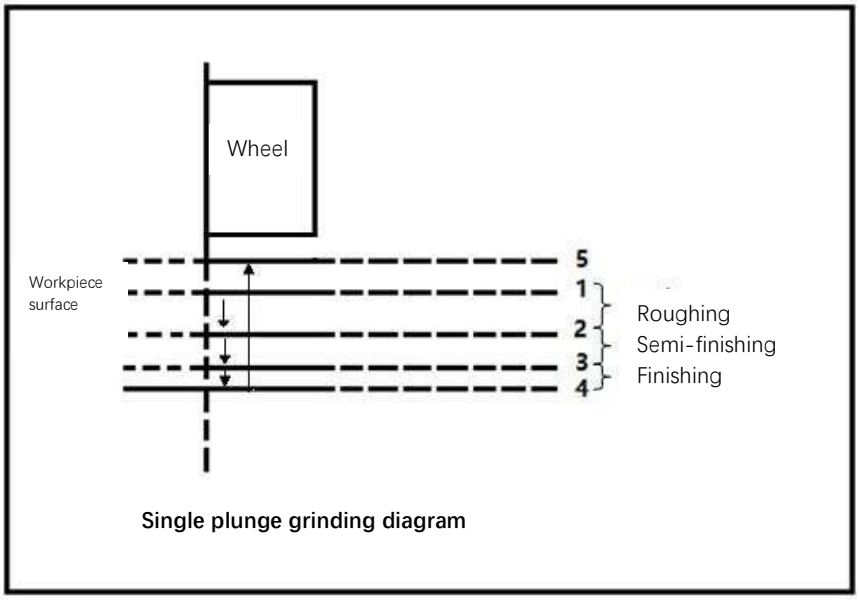
A — A: When no value is specified, establish workpiece coordinate system using T0101. When the value is 1, establish workpiece coordinate system using G54 mode.

O — Whether the coordinate system is corrected after the measuring instrument dimension arrival signal is triggered. When no value is specified, coordinates are not corrected after the dimension is in position; when 1 is specified, this function is enabled. It is used to dress other external circles with external circle of the measuring instrument as the reference coordinate system

Note

I. For the situation that measuring instrument is used, when the corresponding technology in-position signal is detected during G77 execution, the block skip is executed to terminate currently executing of G77 and system continues to execute the next line of G code. If G77 is executed when there is measuring instrument failure, when the theoretical size is in position actually, system will continue to perform 1mm more grinding amount and then stop further grinding, and the X axis will move back to the zero point. For the situation that measuring instrument is not used, system performs the feed according to the specified commands.

II. Re-grinding with measuring instrument. During the G77 execution, if there is a corresponding signal of grinding stage after measuring instrument extends, use the X_ command coordinates and speed corresponding to this stage to complete this stage, and then proceed to the next stage of grinding to automatically perform re-grinding. For example, if there is a signal of the P1 roughing stage after measuring instrument extends, use the X coordinates and feed rate of the finishing stage to complete the roughing, and perform G77X_P2 of semi-finishing stage.



Single plunge grinding diagram

Path of grinding wheel in cross grinding (single-time plunge grinding)

(1) The grinding wheel moves to original grinding position 1;

(2) Rough grinding: The grinding wheel moves from position 1 to position 2;

(3) Semi-fine grinding: The grinding wheel moves from position 2 to position 3;

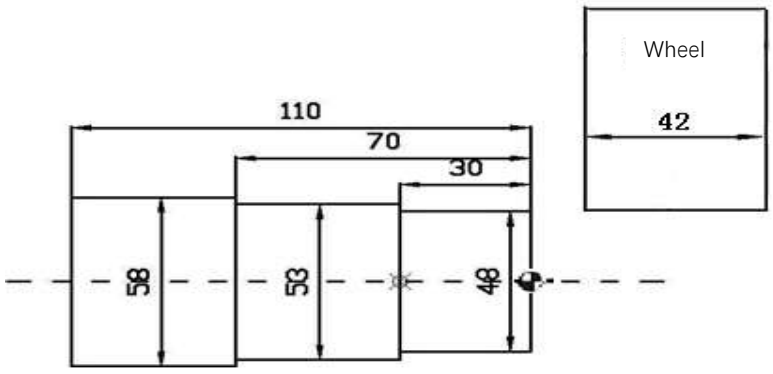
(4) Fine grinding: The grinding wheel moves from position 3 to position 4;

(5) Polishing: The grinding wheel stays at position 4;

(7) Grinding wheel retracts: The grinding wheel moves from position 4 to position 5.

Example

As shown below. The workpiece has three plummer blocks, which include plummer block 1, plummer block 2 and plummer block 3 from right and left. Plummer block 1: workblank diameter is 48.2mm, final diameter is 48mm, length is 30mm; plummer block 2: workblank diameter is 53.1mm, final diameter is 53mm, length is 40mm; plummer block 3: workblank diameter is 58.3mm, final diameter is 58mm, length is 40mm. Width of grinding wheel is 42mm.



%0709;;	
M46;	Start the oil pump
M33	Start the grinding wheel

M7;	Enable cooling
M3 S800	Start the headstock
Determine whether the grinding wheel needs dressed	
IF #54100 EQ #54101	Determine whether the grinding wheel needs dressed
T0102	Confirm to dress coordinate system of the grinding wheel and select offset of tool 2
G1 X10 F1000	The diamond pen nib is 10mm from outside edge of the grinding wheel
Z0	The diamond pen nib is 0mm from end face of the grinding wheel
G1 X1	Move to the position 1mm in rapid traverse
G1 X0 F100	G1 mode, low-speed feed to X0 position
G79I-0.05 Z43 E100F400 N2	Call G79 dressing wheel canned cycle
G1 X10 F100	The grinding wheel retracts to X10 position
#54000=0	The auxiliary variable of wheel dressing: number of grinding workpieces is cleared to enter the next cycle determination
ENDIF	
Start workpiece machining, grinding of plummer block 1	
T0101	Confirm the workpiece coordinate system
G1 Z0 F1000	The grinding wheel moves to Z0 end face of the workpiece coordinate system
G1 X49 F500	Move to diameter X49 at G1 speed
G1 X48.2 F10	Move to the workpiece surface at low speed
G1 X48.15 F0.5	Grind to X48.15 and the measuring instrument stretches
M12	The measuring instrument stretches
G77 X48.1 F0.4 P1	G77 cross grinding canned cycle--rough grinding
G77 X48.05 F0.4 P2	G77 cross grinding canned cycle--semi-fine grinding
G77 X48.02 F0.2 P3	G77 cross grinding canned cycle--fine grinding
G77 X48 F0.1 P4	G77 cross grinding canned cycle--polishing
G04 X1	Dwell 1S
M13	The measuring instrument retracts
G1 X80 F1000	The grinding wheel retracts to X80
Plummer block 2	
Z-40	Z moves right to Z-70, start grinding plummer block 2
X53.3	The wheel moves to the diameter X53.5 position in rapid traverse
X53.2 F50	Move close to the blank surface
X53.1 F5	Move to the blank surface at low speed
G1X53.05F0.4	Rough grinding
G1X53.02F0.2	Semi-fine grinding

G1X53F0.1	Fine grinding
G04X2	Dwell 2s, polishing
G1X80F1000	The grinding wheel retracts
Plummer block 3	
Z-80	Z moves right to Z-80, start grinding plummer block 3
X58.3 F100	Move close to the blank surface
G1X58.1 F5	Move to the blank surface at low speed
G1X58.05 F0.4	Rough grinding
G1X58.02 F0.2	Semi-fine grinding
G1X58F0.1	Fine grinding
G04X2	Pause 2s, polishing
G53G0X0	Return to zero point on X axis
#54000=#54000+1	Auxiliary wheel dressing: the number of ground workpieces is increasing successively
M30	

Multi-times plunge grinding: G77 X (U) _ F_ P_ A_ O_ B_ C_ W_

X (U) — X is the target coordinate value. G90 is the absolute programming mode, and G91 is the incremental programming mode. U is a value relative to start point of grinding, and ± represents directions. (Diameter programming is by default. If it needs to be modified, modify diameter/radius mode in system parameters)

F — Feedrate of grinding wheel (X direction).

P — Measuring instrument signal. If P value is set, it means that the measuring instrument is used; otherwise, it means that measuring instrument is not used. P1 represents the rough grinding in-position signal. P2 represents the semi-fine grinding in-position signal. P3 represents the fine grinding in-position signal. P4 represents the dimension signal.

A — A: When no value is specified, establish workpiece coordinate system using T0101. When the value is 1, establish workpiece coordinate system using G54 mode.

O — Whether the coordinate system is corrected after the measuring instrument dimension arrival signal is triggered. When no value is specified, coordinates are not corrected after the dimension is in position; when 1 is specified, this function is enabled. It is used to dress other external circles with external circle of the measuring instrument as the reference coordinate system.

B — Width of grinding wheel (mm). When no value is specified, this function is invalid.

C — The amount of overlap of the grinding wheel movement on Z (unit: mm). When no value is specified, this function is invalid.

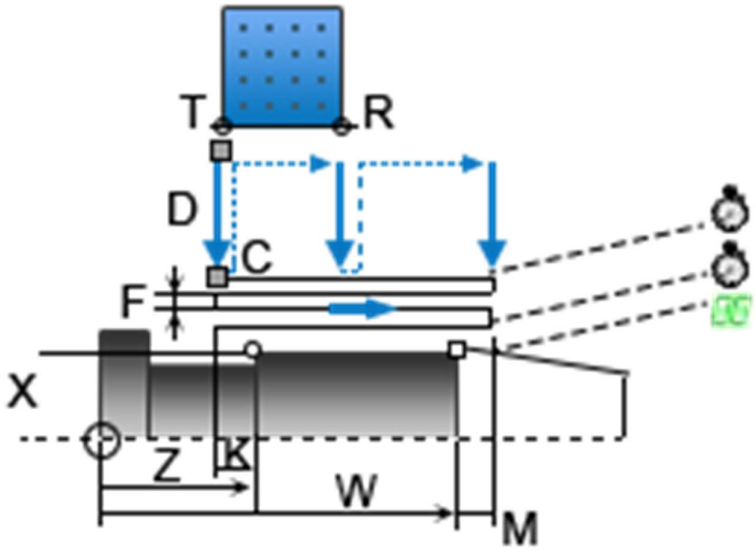
W — Target coordinate on Z. It is the value relative to the grinding start point. ± represents the movement direction on Z. When no value is specified, this function is invalid.

Note:

I. For the situation that measuring instrument is used, when the corresponding technology in-position signal is detected during G77 execution, the block skip is executed to terminate currently executing of G77 and system continues to execute the next line of G code. If G77 is executed when there is measuring instrument failure, when the theoretical size is in position actually, system will continue to perform 1mm more grinding amount and then stop further grinding, and the X axis will move back to the zero point. For the situation that measuring instrument is not used, system performs the feed according to the specified commands.

II. Re-grinding with measuring instrument. During the G77 execution, if there is a corresponding signal of grinding stage after measuring instrument extends, use the X_ command coordinates and speed corresponding to this stage to complete this stage, and then proceed to the next stage of grinding to automatically perform re-grinding. For example, if there is a signal of the P1 roughing stage after measuring instrument extends, use the X coordinates and feed rate of the finishing stage to complete the roughing, and perform G77X_P2 of semi-finishing stage.

III. After multi-time plunge grinding is completed, the grinding start point on Z is returned to completed G77, and execute the next line of G codes.



Path of grinding wheel in multi-times plunge grinding (longitudinal grinding)

(1) G77 canned cycle, the grinding wheel moves to original grinding position X_D;

(2) G77 canned cycle, the grinding wheel moves from position X_D to position X_C for the first time;

(3) G77 canned cycle, the grinding wheel moves from position X_D to position X_C many times;

(4) G76 canned cycle, the grinding wheel moves from position Z_D to position Z_M.

Example

T0101	Determine the workpiece coordinate system
G1 Z0 F1000	The wheel moves to the Z0 end face position of the workpiece coordinate system
G1 X49 F500	Move to X49 ast G1 speed
G1 X48.5 F50	Move close to the blank surface

G1 X48.2 F2	Move to the blank surface at low speed
G1 X48.15 F0.5	Grind to X48.15, and extend the measuring instrument
M12	Extend the measuring instrument
G77X48.1 W100 B50 C10 F0.4 P1	G77 grind multiple times to the rough grinding signal stage
G76 X48.05 Z100 R0.01 E0.2 F400 P2	G76 semi-fine grinding
G76 X48.02 Z100 R0.002 E0.1 F400 P3	G76 fine grinding
G76 X48. Z100R0.001 E0.1 F400 P4	G76 polishing
G04 X1	Dwell 1 second
M13	Measuring instrument returns
G1 X80 F1000	The wheel retracts to X80
M30	

G77 multi-times plunge grinding+G76 longitudinal grinding canned cycle greatly simplifies machining program, and G31 block-skip is not needed

3.1.2 Longitudinal Grinding

G76 X（U）_ Z(W)_ R_ E_ F_ P_ A_ D_ O_ Q_

X (U) — X is the target coordinate value. G90 is the absolute programming mode, and G91 is the incremental programming mode. U is a value relative to start point of grinding, and ± represents directions. (Diameter programming is by default. If it needs to be modified, modify diameter/radius mode in system parameters)

Z (W) — The target coordinate value on Z, the distance the wheel moved in Z direction. G90 is the absolute programming mode, and G91 is the incremental programming mode. W is a value relative to start point of grinding, and ± represents directions. (It is best to take the grinding starting point of the edge of the workpiece as the datum)

R — Grinding amount. Unit: mm. Generally, parameters can be set as follows: rough grinding: 0.05; semi-fine grinding: 0.03; fine grinding: 0.02, polishing: 0.01. Specific setting should be based on customer's technological requirements.

E — Feedrate of grinding wheel (X direction).

F — Feedrate of grinding wheel (Z direction).

P — Measuring instrument signal. When P value is specified during programming, it means that the measuring instrument is used; otherwise, it means that the measuring instrument is not used.

P1 represents the rough grinding in-position signal. P2 represents the semi-fine grinding in-position signal. P3 represents the fine grinding in-position signal. P4 represents the final dimension arrival signal.

A — When no value is specified, establish workpiece coordinate system using T0101. When the value is 1, establish workpiece coordinate system using G54.

D — When no value is specified, bilateral feed is by default; when 1 is specified, it means lateral feed.

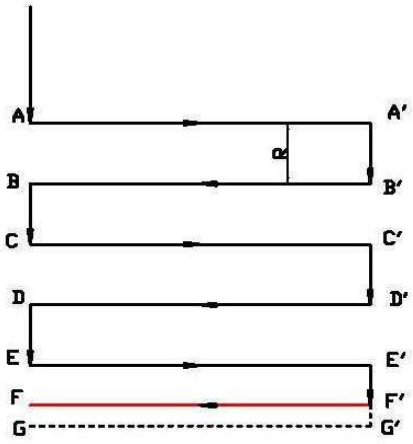
O — Whether the coordinate system is corrected after the measuring instrument dimension arrival signal is triggered. When no value is specified, coordinates are not corrected after the dimension is in position; when 1 is specified, this function is enabled. It is used to dress other external circles with external circle of the measuring instrument as the reference coordinate system.

Note

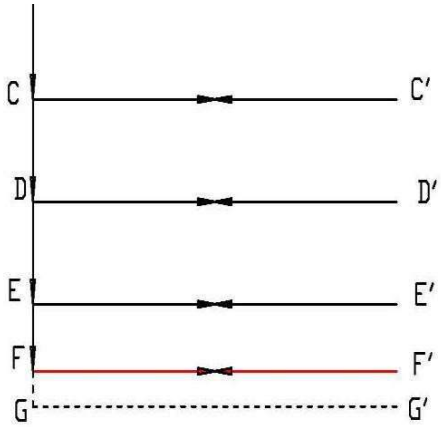
I. For the situation that measuring instrument is used, when the corresponding technology in-position signal is detected during G76 execution, the block skip is executed to terminate currently executing of G77 and system continues to execute the next line of G code. If G76 is executed when there is measuring instrument failure, when the theoretical size is in position actually, system will continue to perform 1mm more grinding amount and then stop further grinding, and the X axis will move back to the zero point. For the situation that measuring instrument is not used, system performs the feed according to the specified commands.

II. Re-grinding with measuring instrument. During the G76 execution, if there is a corresponding signal of grinding stage after measuring instrument extends, use the X_ command coordinates and speed corresponding to this stage to complete this stage, and then proceed to the next stage of grinding to automatically perform re-grinding. For example, if there is a signal of the P1 roughing stage after measuring instrument extends, use the X coordinates and feed rate of the finishing stage to complete the roughing, and perform G77X_P2 of semi-finishing stage.

III. For the situation that measuring instrument is used, when the bilateral grinding is executed, in order to ensure the last step of feed at the measuring instrument end and prevent the final size from being small, the G76P4 polishing stage is of the lateral grinding mode.



Bilateral grinding

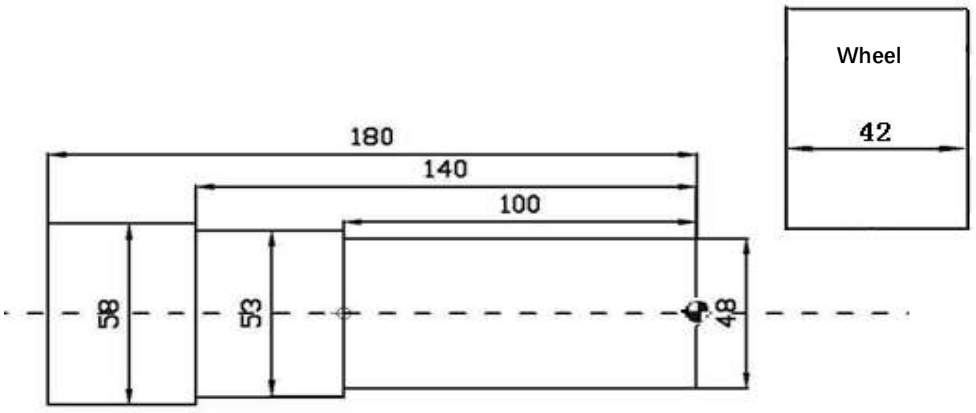


Lateral grinding

In the above figure, FF' represents the final size arrival, namely the measuring instrument signaled size reaches the skip position of G31

Example

As shown below. The workpiece has three plummer blocks s, which include plummer block 1, plummer block 2 and plummer block 3 from right and left. Plummer block 1: workblank diameter is 48.2mm, final diameter is 48mm, length is 100mm; plummer block 2: workblank diameter is 53.1mm, final diameter is 53mm, length is 40mm; plummer block 3: workblank diameter is 58.3mm, final diameter is 58mm, length is 40mm. Width of grinding wheel is 42mm.



Grinding wheel

Programming example

%0709;;	
M46;	Start the oil pump
M33;	Start the grinding wheel
M7	Enable cooling
M3S800	Start the headstock
Determine whether the grinding wheel needs dressed	
IF #54100 EQ #54101	Determine whether the grinding wheel needs dressed
T0102	Confirm to dress coordinate system of the grinding wheel and select offset of tool 2
G1 X10 F1000	The diamond pen nib is 10mm from outside edge of the grinding wheel
Z0	The diamond pen nib is 0mm from end face of the grinding wheel
G1 X1	Move to 1mm position in rapid traverse
G1 X0 F100	G1 mode, feed to X0 position at low speed
G79 I-0.05 Z45 E100 F400 N2	Call G79 dressing wheel canned cycle
G1 X10 F100	The grinding wheel retracts to X10 position
#54100=0	The auxiliary variable of wheel dressing: number of grinding workpieces is cleared to enter the next cycle determination
ENDIF	
Start workpiece machining, grinding of plummer block 1	
T0101	Confirm the workpiece coordinate system
G1 Z0 F1000	The grinding wheel moves to Z0 end face of the workpiece coordinate system
G1 X49 F500	Move to diameter X49 at G1 speed
G1 X48.4	
G1 X48.2 F2	Move to the workpiece surface at low speed

G1 X48.17 FO.5	Grind to X48.17 and the measuring instrument stretches
W-58	
W58	
M12	The measuring instrument extends
G76 X48.1 Z-58 R0.01 E0.4 F400 P1	G76 rough grinding, the tool movement on Z is the workpiece size minus the wheel width
G76 X48.05 Z-58 R0.01 E0.2F400 P2	G76 semi-fine grinding
G76 X48.02 Z-58 R0.005 E0.1F400 P3	G76 fine grinding
G76 X48 Z-58 R0.005 E0.1 F400 P4	G76 polishing
G04 X1	Dwell 1S
M13	The measuring instrument retracts
G1 X80 F1000	The grinding wheel retracts to X80
Plummer block 2	
Z-140	Z moves right to Z-140 position, start grinding plummer block 2
X53.3	
X53.2 F10	
X53.1 F2	
G1 X53.05 F0.4	Rough grinding
G1 X53.02 F0.2	Fine grinding
G1 X53 F0.1	Polishing
G04 X2	Dwell 2s, the size arrives
G1 X80 F1000	The grinding wheel retracts
Plummer block 3	
Z-180	Z moves right to Z-180 position, start grinding plummer block 3
X58.3	
G1 X58.3F100	
G1 X58.1 F2	
G1 X58.05 F0.4	Rough grinding
G1 X58.02 F0.2	Fine grinding
G1 X58 F0.1	Polishing
G04 X2	Dwell 2s, the size arrives
G53 G1 X0 F1000	Return to zero point on X
#54100=#54100+1	Auxiliary wheel dressing: the number of ground workpieces is increasing successively
M30	

3.1.3 Grinding Wheel Dressing

G79 I_ Z_E_F_N _Q_A_D_

Interpretation:

I — Single dressing of grinding wheel. Generally the I value is negative.

Z — Coordinate value. Generally it refers to movement distance in Z direction (exceeding thickness of grinding wheel). Z may be positive or negative. Actually, displacement distance in Z direction should add up to thickness ±3MM of grinding wheel in the grinding system.

E — Feedrate of grinding wheel in X direction.

F — Movement speed of worktable in Z axis.

N — Grinding wheel dressing times.

Q — Whether returns to the starting point of grinding after program ends.

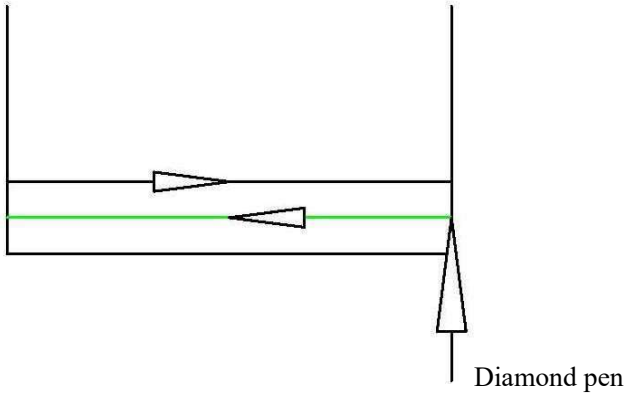
When no value is specified: Yes;

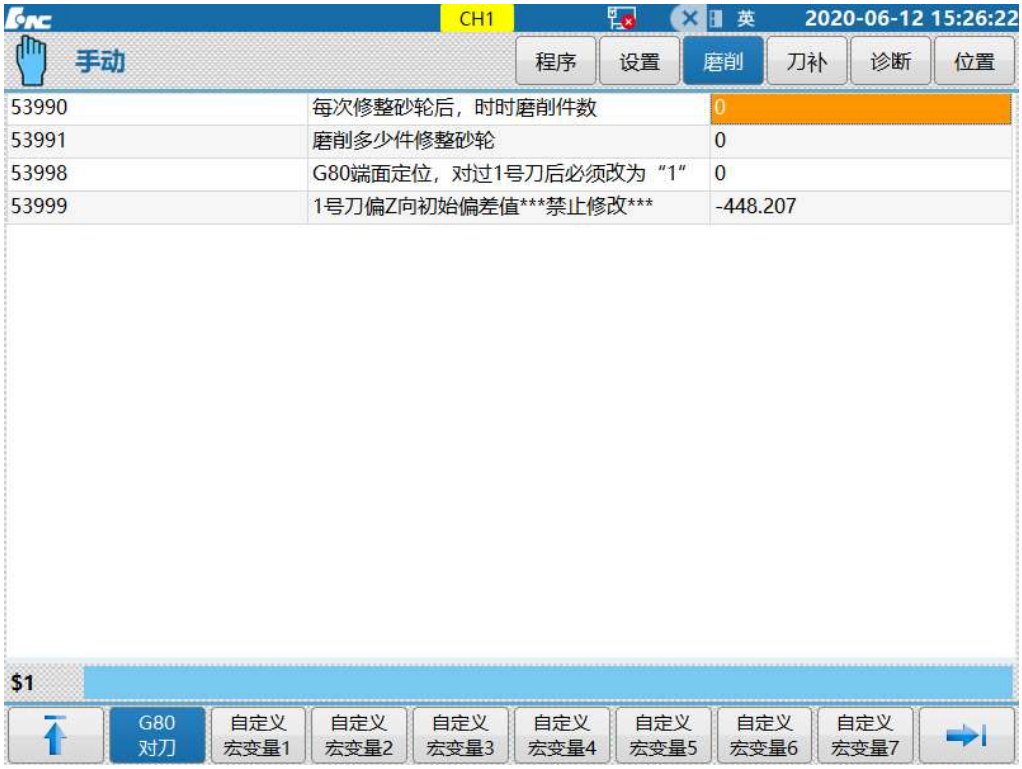
when 1 is set: No.

A — A: When no value is specified, establish workpiece coordinate system using T0102. When the value is 1, establish workpiece coordinate system using G54 mode.

D — Lateral/bilateral feed selection. Bilateral by default. 1: Lateral feed.

K — Dressing composition coefficient. The coordinate system compensation amount is equal to the dressing amount multiplied by the coefficient; when 1 is specified, the workpiece coordinate system is established in G54 mode.





Note

I. During wheel dressing, I and F, N values can be set to determine whether to use the roughing or finishing.

II. The number of grinding workpieces can be set to reach the set value for grinding wheel dressing. Variable #53990 records the number of workpiece grinding times, #53991 dressing the grinding wheel every time how many workpieces are ground, and when #53990 is equal to #53991, the grinding wheel dressing is performed

Example

Program of grinding wheel dressing

%5930	
IF [#53990 EQ #53991]	Wheel dressing will be performed if workpiece grinding times reaches the set value
T0102	Conform the coordinate system of grinding wheel dressing
M33	Enable the grinding wheel
M7	Turn on cutting fluid
G1 X10 F500	The grinding wheel moves to 10mm from diamond pen nib
G1 X2	
G1 X0 F10	The grinding wheel moves to X0
Z0	The worktable moves to Z0
G79 I-0.01 Z55 E10F100 N2	Execute G79 command
G1 X10	Retract
M9	

M34	Disable the grinding wheel
#53990 = 0	Number of ground workpieces is cleared after the wheel is dressed
ENDIF	
M30	

3.1.4 End Face Positioning

G80 W_ E_ F_ H_ J_ A_

W — The distance from the starting point of measurement to the probe (generally it is the tool setting point of workpiece end face measurement T0103 coordinate system) in Z direction while positioning the end face to the fault point of the measuring instrument. Moving this distance in the Z direction must ensure that the measuring instrument signal can be triggered. ± represents the direction of movement in the Z direction.

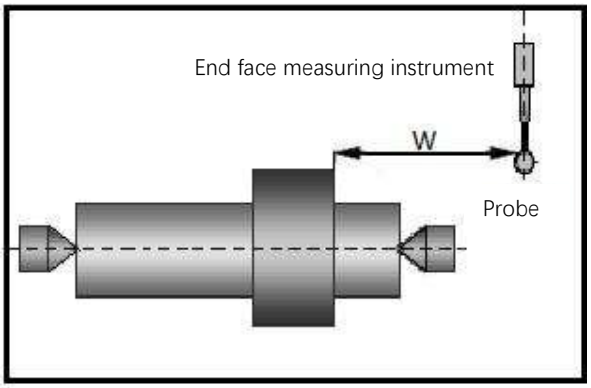
E — The first positioning speed, feedrate on Z.

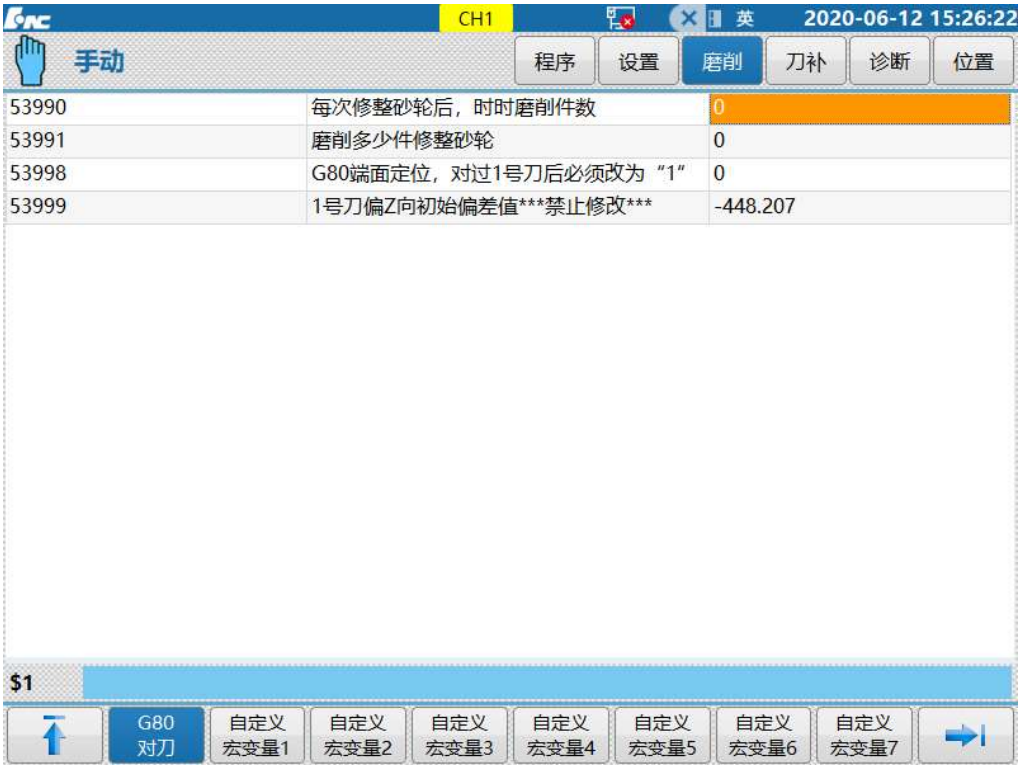
F — The second positioning speed, feedrate on Z.

H — The allowable maximum measurement error. When this value is exceeded, the coordinate system is not performed and the system alarm. When no value is specified, 0.5mm is the default.

J — The retract distance after the measurement instrument signal is triggered for the first positioning. It is a positive value, in the unit mm.

A — A: When no value is specified, establish workpiece coordinate system using T0101. When the value is 1, establish workpiece coordinate system using G5.





For G80 end face positioning, if the zero point of tool 1 (the workpiece grinding coordinate system zero on Z) has been set, #53998 must be changed to 1 manually. Then when G80 is executed for the first time, the system will assign the tool 1 zero point on Z to the #53999 variable and store it as the Z-direction reference; and use the Z-direction position of the second trigger signal of the end-face measuring instrument as the T0103 coordinate system zero on Z, then set #53998 to zero. Later, when executing G80, the difference between the second trigger signal Z-direction position of the end face measuring instrument and the Z-direction zero point of T0103 coordinate system will be superimposed to the reference as the T0101 coordinate system zero on Z.

Note

G80 end face positioning is based on the Z-direction zero point of the first workpiece (blank) T0101 coordinate system as the reference, and the Z-direction zero point of the subsequent workpiece as the reference plus the deviation measurement value. The reference is stored in variable #53999, and the batch processing datum remains unchanged to ensure that the measurement deviation will not accumulate. If the workpieces is changed and tool setting is performed again, due to the reference change, the variable #53998 must be changed to 1 to re-set the reference.

End face positioning program

%111	
T0103	End face measurement coordinate system
M46	Start the oil pump

M16	The measuring instrument stretches
G1 X100 F1000	The grinding wheel moves
Z100	
X50	
Z4	The worktable moves to the right position
G80 W-5 E50 F5 H1 J0.5	Call G80
M17	The measuring instrument retracts
G1 X100 F1000	
M30	

3.1.5 Wheel End face Dressing

G81 X(U)_ Z(W)_ E_ F_ N_

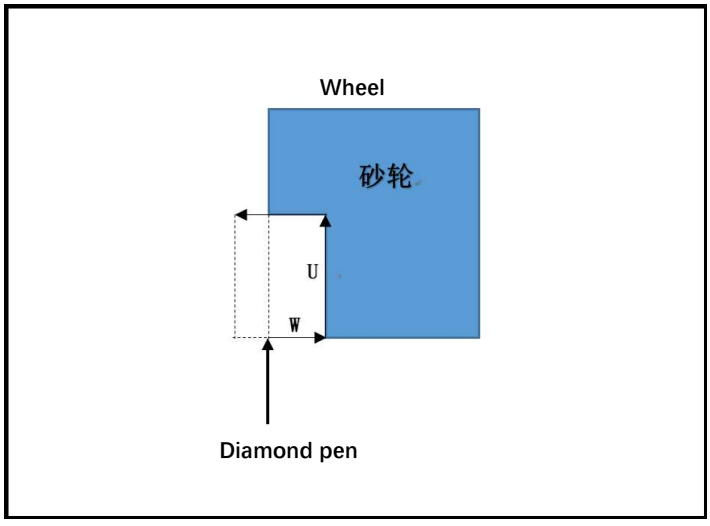
X (U) -- Dressing depth on X

Z (W) -- Feed amount on Z

E -- Feedrate on X

F -- Feedrated on Z

N -- Dressing times. When no value is specified, the default is 1.



Note

①The coordinate system of grinding wheel end face dressing is T0104.

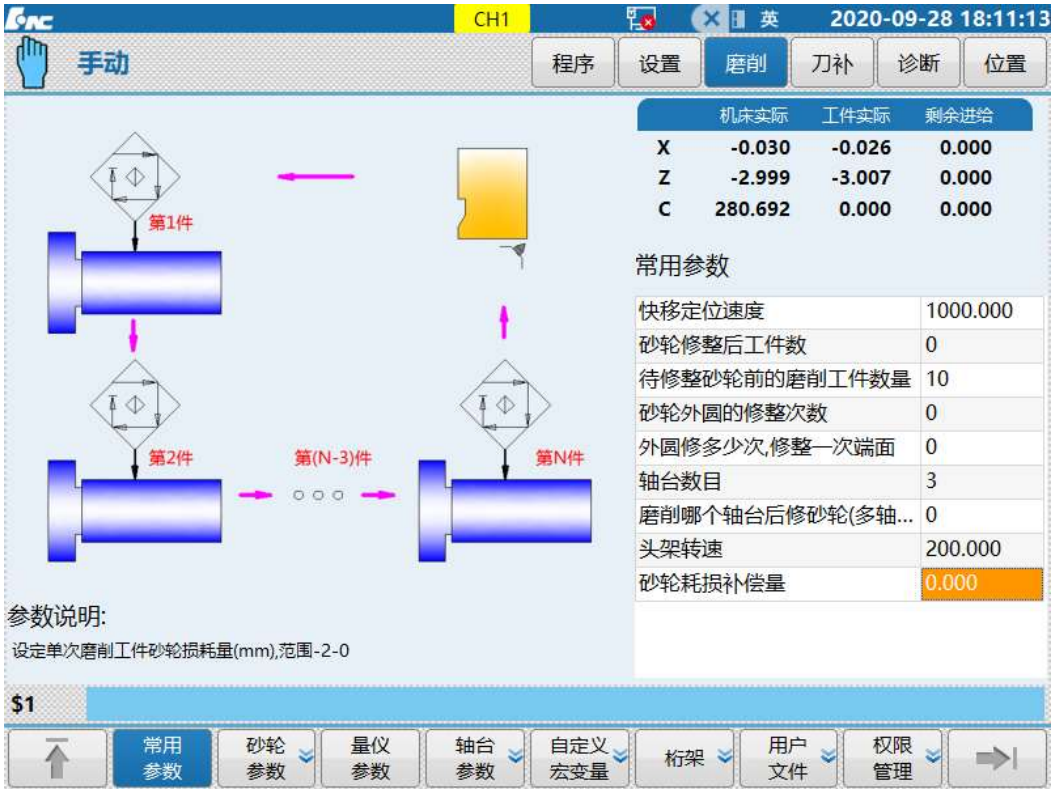
② When using end face grinding, the wear of the end face of the grinding wheel is lower than that of the outer circle, and the frequency of dressing the end face is generally lower than that of the outer circle. Users can set the end face dressing after the outer circle dressing reaches the set number of times.

Example

%1234	
T0104	The wheel end face dressing coordinate system
M33	Enable the wheel
M7	Enable the cutting fluid
G1 X10 F500	The wheel moves to the 10mm position from diamond pen nib
G1 X2	
Z0	Worktable moves to Z0
G1 X0 F10	Wheel moves to X0
G81 U30 W0.02 E50 F10	Execute G81
G1 X10	Retract
M9	
M34	Disable the wheel
M30	

3.2 Process Card

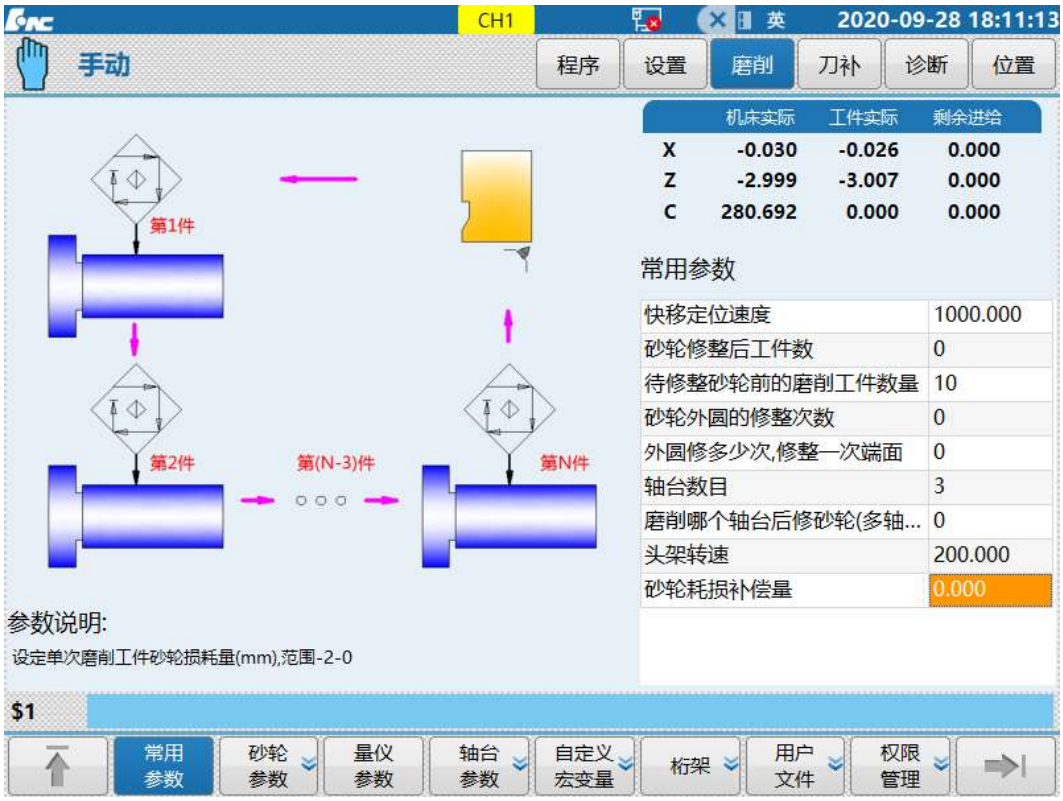
3.2.1 Commonly Used Parameters



Commonly used parameter list:

Contents of parameters	Setup description
Rapid traverse positioning speed	Feedrate in rapid traverse positioning, unit: mm/min
Number of ground workpieces after wheel dressing	The initial value is 0. The number if ground workpieces after wheel dressing. It is cleared automatically after dressing.
Number of ground workpieces before wheel dressing	Number of the ground workpieces. When the actual number reaches the set value, the wheel will be dressed automatically.
Number of dressing of the wheel outer-circle	Number of times that the outer circle of wheel is dressed. It is set to 0 after wheel end face is dressed
How many times the outer circle of the grinding wheel is dressed every time the end face is dressed	To set How many times the outer circle of the grinding wheel is dressed every time the end face is dressed
Number of plummer blocks	Set the number of plummer blocks to be ground, 4 at most
Headstock speed	Set the speed of servo headstock and frequency conversion headstock. Unit: r/min
Wheel wear compensation amount	Set the wear of wheel for single grinding (mm), ranges from -2 to 0

3.2.2 Parameters Of Grinding Wheel



Basic parameters:

Contents of parameters	Setup description
Grinding wheel width	Width of grinding wheel. Unit: mm
Wheel real-time diameter	Real-time diameter of wheel. Unit: mm
Minimum failure diameter of wheel	The minimum diameter of wheel. Unit: mm
Constant linear speed Enable	0: is disabled; 1: is enabled. Frequency conversion wheel is valid
Constant linear speed (m/s)	Unit: m/s
Speed corresponding to linear speed (r/min)	It is the monitoring value

Dress technology

Contents of parameters	Setup description
Whether to dress the grinding wheel	0：dress the wheel automatically in machining；1：dress the wheel seperately
Dressing direction	-1: from left to right; 1: from right to left. Determine based on the first tool moving direction
Lateral/Bilateral dressing	0: lateral；1: bilateral. Note: it must be 1 when dressing the end face
Rapid safe position on X	Generally can be set as 1 to 5mm
Rapid safe position on Z	Defaulted 1mm. Set based on the actual requirements
Dressing start point return speed	From the safe position set above to the position of X0Z1 or X0Z-1, the corresponding F value
X-direction distance of start point return	Retract distance on X after dressing, diameter, unit: mm. The lateral dressing is valid.
Single-step amount in roughing on X	Set based on the actual requirements
Roughing speed on X	
Roughing speed on Z	
Roughing times	
Single-step amount in finishing	Set based on the actual requirements

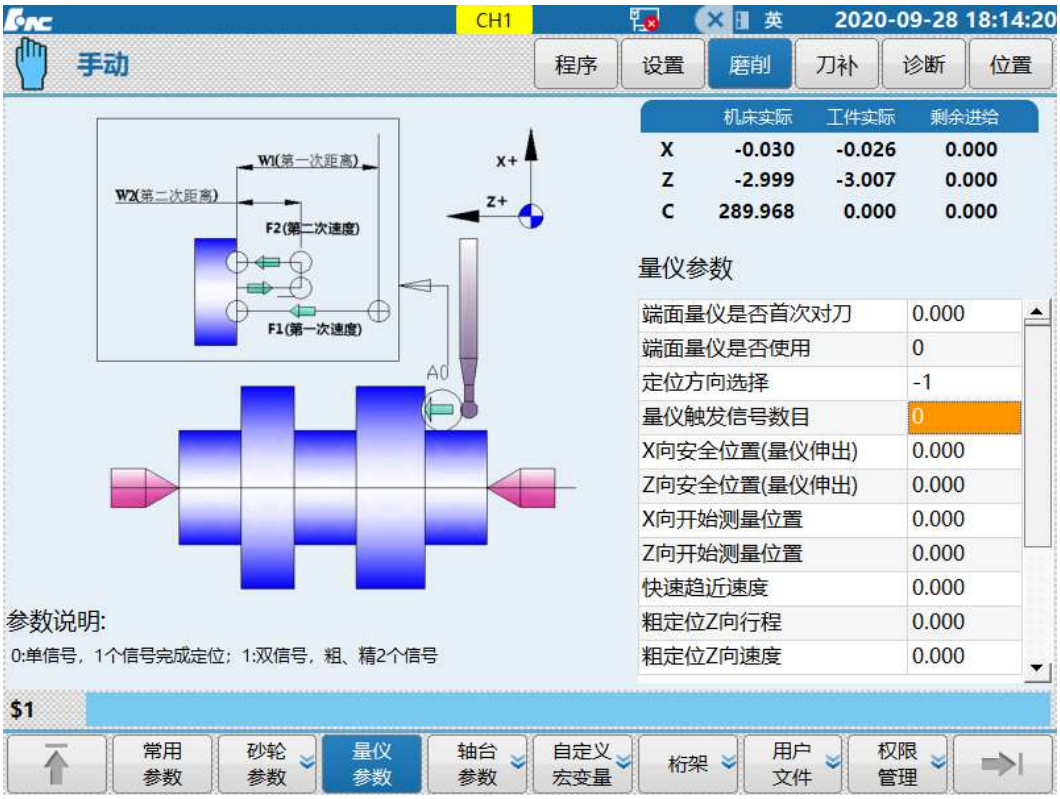
on X	
Finishing speed on X	
Finishing speed on Z	
Finishing times	
coordinate compensation coefficient after wheel dressing	1 is the default. Set based on the actual requirements. Range: 0 to 1; if 0 is set, then the coordinate offset will not be performed after dressing
Start point return speed in lateral dressing	Unit: mm/min

End face dressing

Contents of parameters	Setup description
Left end face arc	0: Exist; 1: Not exist
Left end face R	Radius of left end face arc
Left end face dressing selection	0: Not dress; 1: Dress
Left end face dressing amount on Z	Dressing amount of left end face on Z. Unit: mm
Left end face dressing depth on X	Dressing depth of left end face on X. Unit: mm

Right end face arc	0: Exist; 1: Not exist
Right end face R	Radius of right end face arc
Right end face dressing selection	0: Not dress; 1: Dress
Right end face dressing amount on Z	Dressing amount of right end face on Z. Unit: mm
Right end face dressing depth on X	Dressing depth of right end face on X. Unit: mm
End face dressing speed	Unit: mm/min

3.2.3 Parameters of Measuring Instrument

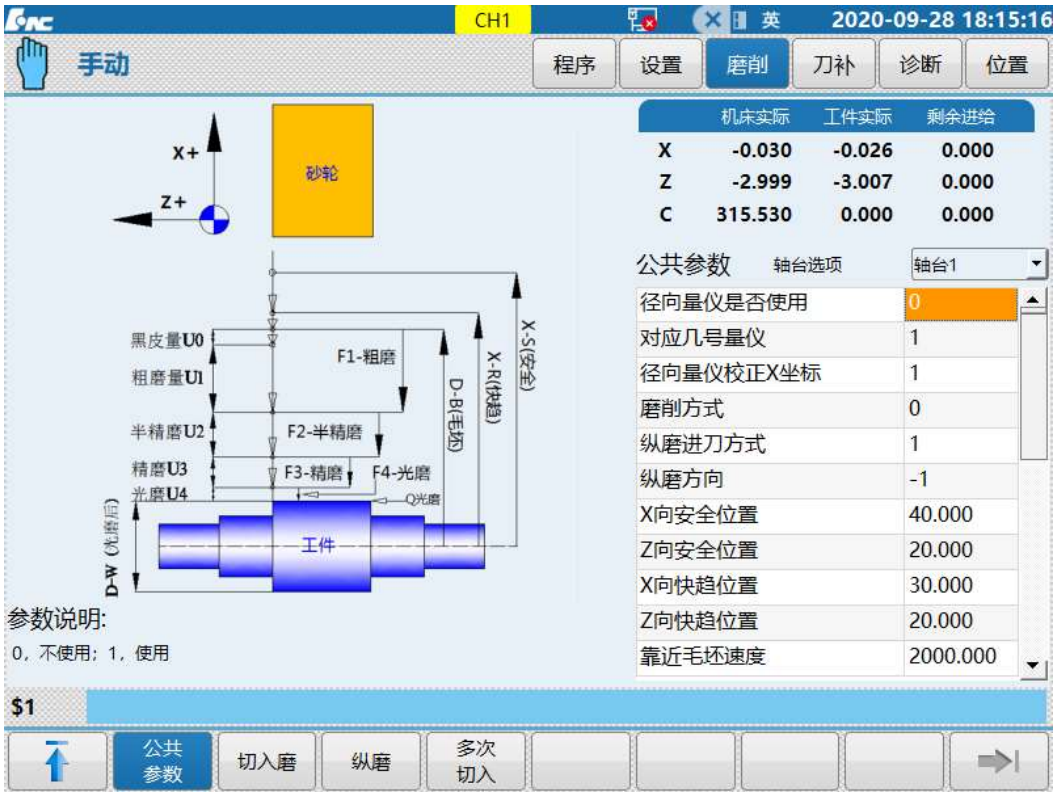


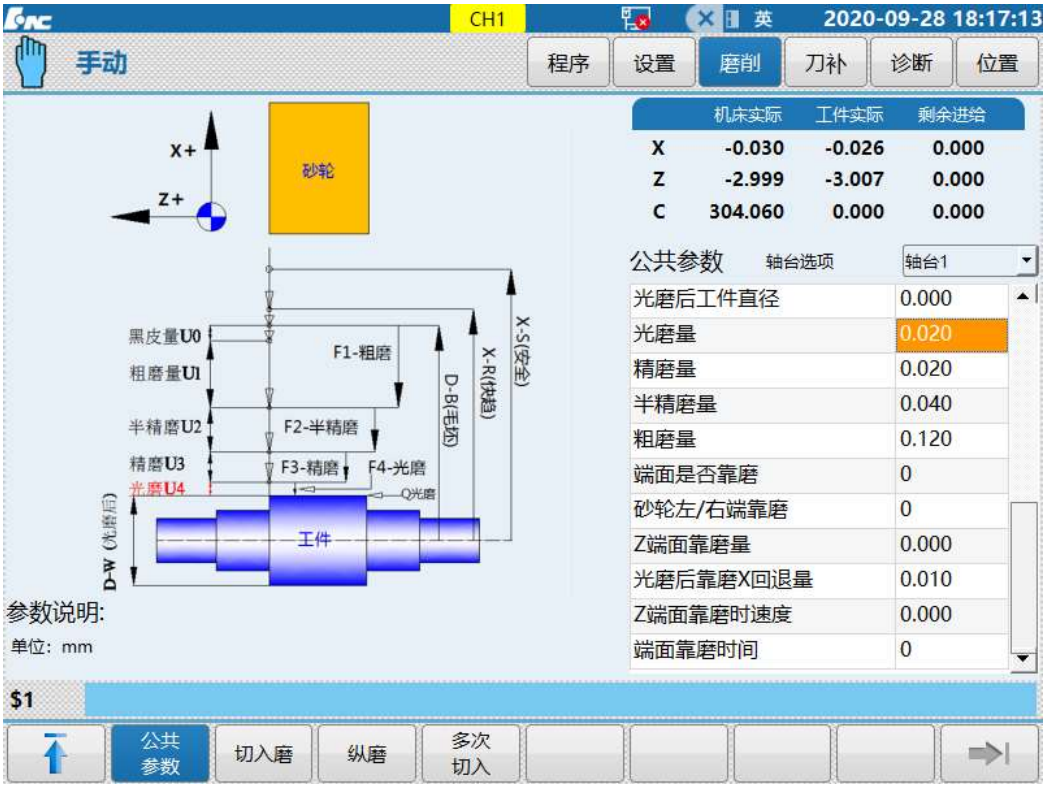
Parameter list of measuring instrument:

Contents of parameters	Setup description
Whether tool setting of measuring instrument is for the first time	0: No; 1: Yes. As long as the Z axis of T0101 performs tool setting again, this parameter needs to be manually set to 1
Whether end face measuring instrument is used	0: No; 1: Yes
The probe is on the left/right of the reference level	-1: Probe is on the right of positioning level; 1: Probe is on the left of positioning level

Safe position in X direction (the measuring instrument stretches)	Position where the measuring instrument stretches. Ensure no collision of measuring instrument
Safe position in Z direction (the measuring instrument stretches)	Position where the measuring instrument stretches. Ensure no collision of measuring instrument
Starting point of measurement in X direction	Starting position of probe measuring
Starting point of measurement in Z direction	Starting position of probe measuring
Rapid approach speed	Speed from safe position to measurement position
Rough positioning travel on Z	Larger than the fine positioning travel
Rough positioning speed on Z	Generally, 10 to 20
Fine positioning travel on Z	Smaller than the first Z value
Fine positioning speed on Z	Round F5. This value can affect the measuring precision
Dwell time after signal trigger (ms)	Set based on the actual requirements. Unit: ms. Generally 0 is set
Allowable maximum measuring error	Compensation is not performed when this error is exceeded
Initial offset on Z	It is generated by automatic measurement, and cannot be changed

3.2.4 Parameters Of Plummer Block

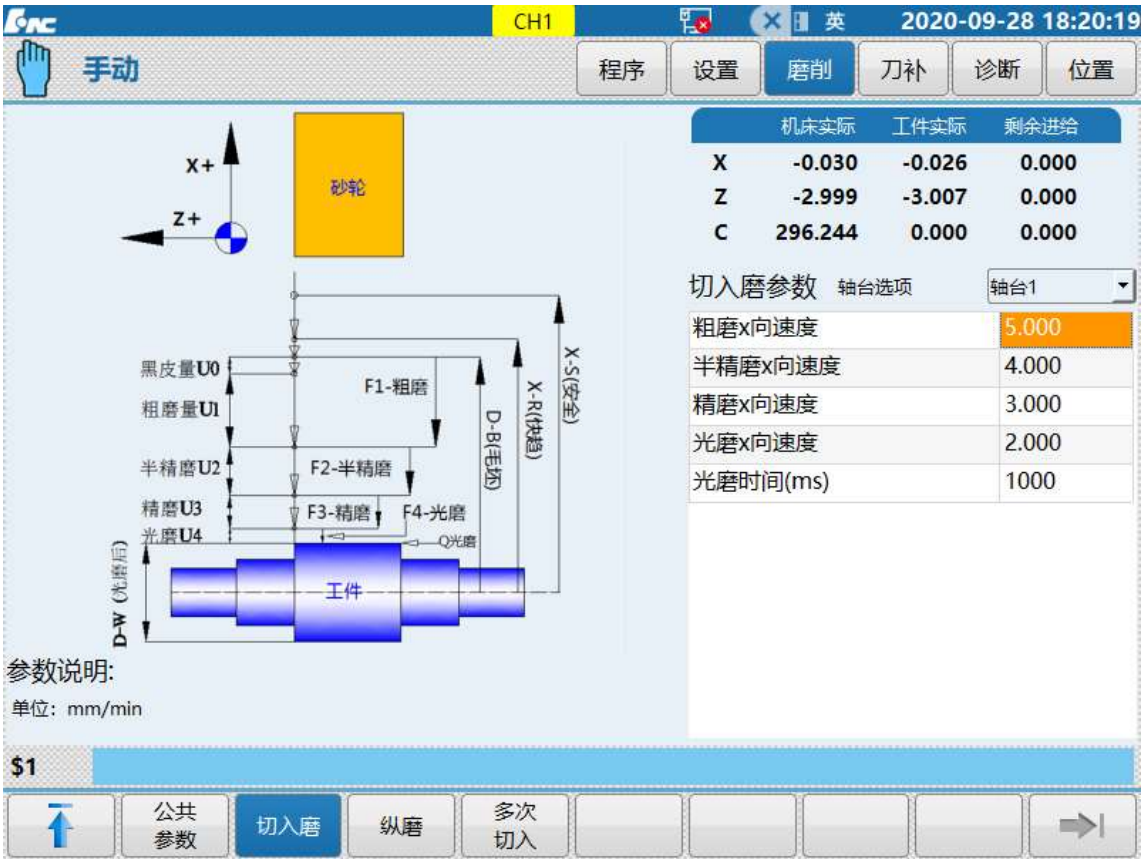




Public parameters

Contents of parameters	Setup description
Whether radial measuring instrument is used	0: No; 1: Yes
Corresponding measuring instrument	1: Radius measuring instrument 1; 2: Radius measuring instrument 2
Radial measuring instrument correction X coordinate	Coordinate origin is corrected based on P4 signal. 0: Not correct; 1: Correct
Grinding mode	0: Plunge grinding; 1: Longitudinal grinding; 2: Multi-times plunge grinding
Safe position on X	Safe position on X
Safe position on Z	Safe position on Z
Rapid approach position on X	Approach workpiece position on X
Rapid approach position on Z	Approach workpiece position on Z
Approaching workblank speed	F value corresponding to G1 from Xpos and Zpos to workblank
Workblank diameter	Grinding diameter, workblank, roughing, semi-finishing, and finsihing are values under diameter programming mode
Grinding amount before measuring instrument stretches	
Workpiece diameter after polishing	
Polishing amount	
Finishing amount	

Semi-finishing amount	
Roughing amount	
Whether grind end face	0: No; 1: Yes
Left/Right end face grinding of wheel	0: Left; 1: Right (Facing the wheel)
Z end face grinding amount	This parameter is a positive value. The system automatically judges positive and negative.
Retract amount in end face grinding after polishing	For Z end face grinding, this parameter must be set, and can be set as 0.05 to 0.1
Z end face grinding speed	End face grinding speed. A smaller value is recommended. The end face of the right-angle grinding wheel cannot bear too much grinding force
End face grinding time	Unit: ms



Plunge grinding parameter

Contents of parameters	Setup description
Roughing speed on X	Set based on actual requirements. Unit: mm/min
Semi-finishing speed on X	
Finishing speed on X	

Polishing speed on X	
Polishing time (ms)	Unit: ms

Longitudinal grinding parameter:

Contents of parameters	Setup description
Feed mode	1: Lateral; 0: bilateral feed
Longitudinal grinding direction	1: Movement in positive Z; -1: Movement in negative Z
Step width	Under the same diameter, the actual size of plummer block in Z direction, also the drawing size
Single-step amount of roughing	Values calculated according to diameter values in the public parameters
Single-step amount of semi-finishing	
Single-step amount of finishing	
Single-step amount of polishing	
Roughing speed on Z	Set based on actual requirements. Unit: mm/min
Semi-finishing speed on Z	
Finishing speed on Z	
Polishing speed on Z	
Whether to dress the grinding wheel after grinding	Judge whether it is necessary to dress the grinding wheel after roughing, semi-finishing and finishing. For specific value, refer to parameter annotation
Retract distance on X after lateral grinding	Used for lateral grinding. If it is 0, return to the starting point of grinding along Z axis against the surface of the outer diameter ground last time

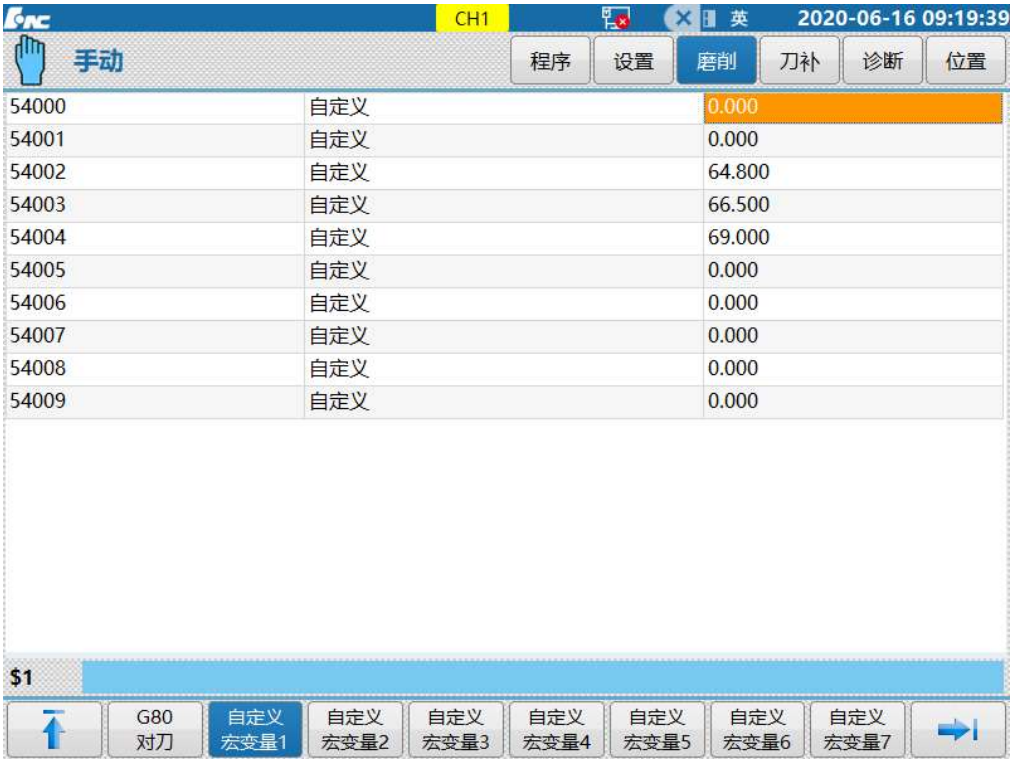
Dwell time Left (ms)	The process requires a pause at both ends for a period of time
Dwell time Right (ms)	
Polishing times	After the set size is reached through polishing, horizontal polishing times in Z direction, and not feed in X direction. If polishing is enabled, it is better to increase polishing size slightly
Overtravel distance Left	The distance that the wheel end face exceeds the plummer block end face to be gound
Overtravel distance Right	

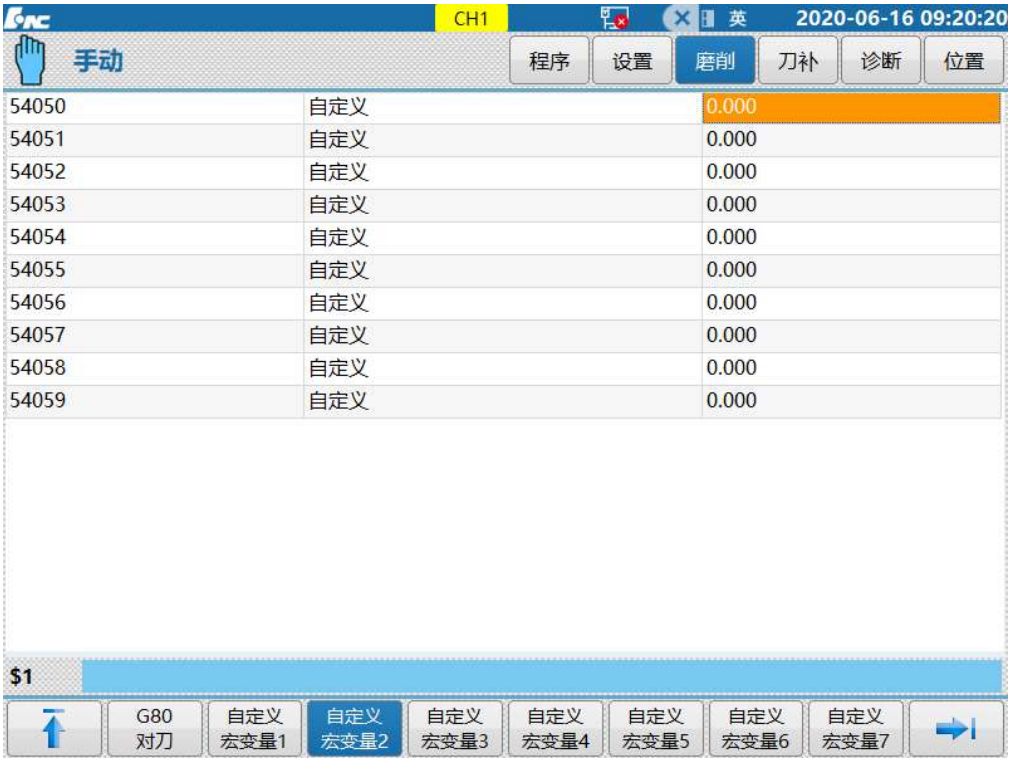
Multi-times plunge grinding parameter

Contents of parameters	Setup description
Stage of plunge grinding	0: Roughing; 1: Semi-finishing; 2: Finishing; 3: Polishing
Plunge grinding mode	Reuse retract amount of crush grinding. Example: Suppose crushing grinding is adopted for coarse grinding and feed grinding is adopted for medium grinding and accurate grinding. Meaning of the parameter: Ensure no collision during crush grinding based on retract amount based on workblank diameter
Z-direction overlap distance	Retract speed

Retract distance after returning to start point	
Retract amount after X single plunge	The retraction amount when plunge grinding is used cyclically. Example: If plunge grinding, semi-finishing, and finishing are used for roughing, and Z-direction grinding is used for polishing, then the meaning of this parameter is: the amount of retraction based on the diameter of the blank to prevent the collision when cutting into the cycle
Retract speed after single plunge	Retract speed
	The value is used for monitoring

3.2.5 Custom Macro Variable





Users can use the custom macro variables provided by the system to program, and can edit the custom macro variables including the types and meanings of the macro variables.

- Customized macro variable editing

(1) Press Set, select Data Manage menu, select User Macro Configuration File under the "Data Management" submenu, insert the U disk, and select USB, as shown in the figure below:

(2) Press the up and down direction keys to confirm the custom macro variable XML configuration file that needs to be edited, and select Load to back up the custom macro variable to the U disk.

Custom macro variables 1 to 9 correspond to USERMACCFG0.XML~USERMACCFG8.XML.

(3) Open the custom macro configuration file with an editing tool for editing

```
<?xml version="1.0" encoding="GB2312"?>
<USERMACCFG version="1.0">

<item no = "54000"name  = "自定义" type= "FLOAT"  min="-9999" max="9999"></item>
<item no = "54001"name  = "自定义" type= "FLOAT"  min="-9999" max="9999"></item>
<item no = "54002"name  = "自定义" type= "FLOAT"  min="-9999" max="9999"></item>
<item no = "54003"name  = "自定义" type= "FLOAT"  min="-9999" max="9999"></item>
<item no = "54004"name  = "自定义" type= "FLOAT"  min="-9999" max="9999"></item>
<item no = "54005"name  = "自定义" type= "FLOAT"  min="-9999" max="9999"></item>
<item no = "54006"name  = "自定义" type= "FLOAT"  min="-9999" max="9999"></item>
<item no = "54007"name  = "自定义" type= "FLOAT"  min="-9999" max="9999"></item>
<item no = "54008"name  = "自定义" type= "FLOAT"  min="-9999" max="9999"></item>
<item no = "54009"name  = "自定义" type= "FLOAT"  min="-9999" max="9999"></item>

</USERMACCFG>

```

no = "54000", the content in " " is the macro variable number; name = "custom", the content in " " is the macro variable meaning;

type= "FLOAT", the content in " " is the macro variable type, FLOAT is a floating-point variable, and INT is an integer variable;

min="-9999", the content in " " is the minimum value of macro variable; max="9999", the content in " " is the maximum value of macro variable.

Note

①The custom macro variables that can be used by users are #53000 to #54999.

②The format of the custom macro variable XML table cannot be changed. The content in " " is the content that can be modified by the user, and other parts are not allowed to be modified.

(4) After configuring the XML form, use the U disk to "load" into the system according to the above operation, and it will take effect after power off and restart.