HC-20XX Manual

（20201017 modification）

(Applicable version V20200924 and above）

Chapter 1 System Introduction

1.1 Button introduction

1.1.1 Single key function

All operations can be realized by single key or combination keys on the panel. The method of using a single key is to press and hold the key until the required function call is completed and then release the key. The mode switch key is valid when the mode switch key is up.

Key name	Function Description
	Auxiliary function shift left.
	Auxiliary function key.
	Auxiliary function key.
	Auxiliary function key.
	Auxiliary function key.
	Auxiliary function shift right.
	In the standby state, the opening and closing of the spindle will automatically turn on when performing automatic processing, and automatically turn off when the end is completed.

	Measuring tool length.
	Perform file management operations, such as loading, copying in, copying out, deleting, etc.
	Enter the function interface such as parameter configuration.
	In the standby state, the A axis moves in the positive direction. Input of number key "1".
	In the standby state, the Y axis moves in the positive direction. Input of number key "2". Select upward in the function options.
	In the standby state, the Z axis moves in the positive direction. Input of number key "3".
	Increase processing speed ratio.
	In the standby state, the X axis moves in the negative direction. Input of number key "4". Select left in the function options.
	In the standby state, the switch between high-speed and low-speed motion during manual operation. Input of number key "5".
	In the standby state, the X axis moves in the positive direction. Input of number key "6". Select right in the function options.
	Switch mobile mode.
	In the standby state, the A axis moves in the negative direction. Input of number key "7".
	In the standby state, the Y axis moves in the negative direction. Input of number key "8". Select down in the function options.
	In the standby state, the Z axis moves in the negative direction. Input of number key "9".
	Reduce processing speed ratio. Input of number key "7".
	Set the current X mechanical coordinate as the X axis workpiece origin. Input of minus sign "-".
	Set the current X mechanical coordinate as the Y axis workpiece origin. Input of number key "3".
	Set the current X mechanical coordinate as the Z axis workpiece origin. Input of decimal point "."
	Set the current X mechanical coordinate as the A axis workpiece origin. Return to the superior interface.

	Empty key, the main key of the combination key.
	Move to X, Y axis the workpiece origin, Z axis raised to a safe height. Confirm operation in function options. Enter and confirm operations.
	Processing start or pause. Modification and deletion of function options.
	Stop current processing. Back to main menu. Exit without saving.

1.1.3 Combination key function

How to use the key combination: hold down the "  " key, press and release other keys, and finally release the "  " key.

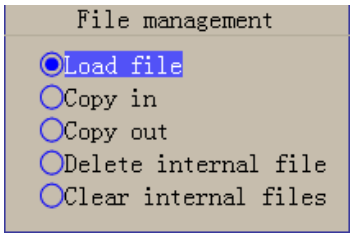
Key 1	Key 2	Function Description
		Spindle speed up 10%.
		Spindle speed down 10%.
		Y axis back to mechanical zero point alone.
		Z axis back to mechanical zero point alone.
		X axis back to mechanical zero point alone.
		A axis back to mechanical zero point alone.
		All axis back to mechanical zero point.
		1.In the high-speed state, set the manual high speed. 2.In the low-speed state, set the manual low speed.
		1.In the high-speed state, set the jog high-speed distance. 2.In the low-speed state, set the jog low-speed distance.
		Enter the advanced processing menu, you can choose "Line number processing", "Breakpoint processing", "Repeated processing", "Array processing".
		Open the tool library settings window.
		Set the tool length of T1.

⇧ Shift	K2	Set the tool length of T1, Unlock the external clamp release button.
⇧ Shift	K3	Set the tool length of T1.
⇧ Shift	K4	Set the tool length of T2.
⇧ Shift	<<	Take a screenshot and save the picture to a U-disk.

Chapter 2 Introduction to Basic Functions

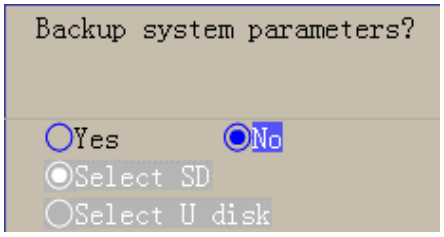
2.1 File management

Press the "  " key to enter the file management interface.



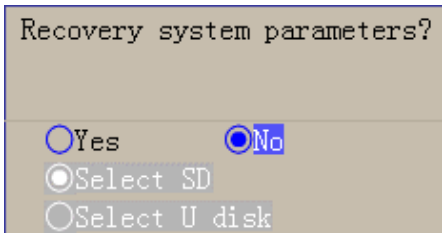
2.2 Backup parameter

Press"  " Key, press "  " key, select "system parameters setting", select "Backup parameter".



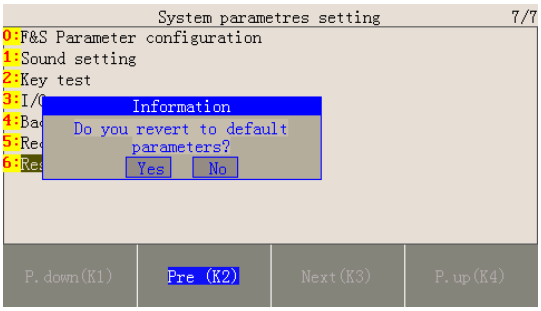
2.3 Recovery parameter

Press"  " Key, press "  " key, select "system parameters setting", select "Recovery parameter".



2.4 Restore to default parameters

Press"  " Key, press "  " key, enter password “6666”, press “  ” key, select "system parameters setting", select "Restore to default parameters".



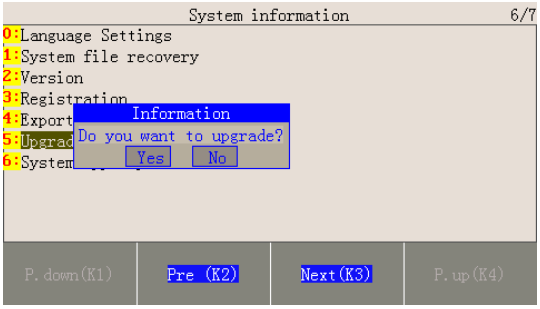
2.5 System Upgrade

There are two ways for the system to enter the upgrade mode to deal with bug fixes in bad situations and to add better functions.

Unzip the upgrade package, copy all the files inside to the root directory of the U disk (without folder), and then insert it into the controller.

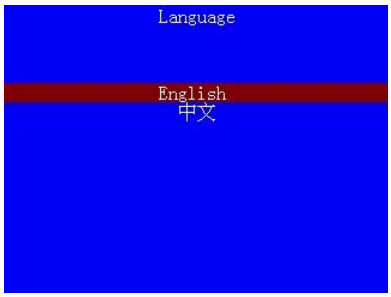
Method 1: Enter "Menu"-"System Information"-" Upgrade", press the " 

" key to enter the upgrade mode.



Method 2: Press and hold the " 

" button, turn on the system, then it will enter the upgrade mode directly.



Select "English"-"Handle", select the ".hcp" file, and press " 

" to upgrade.

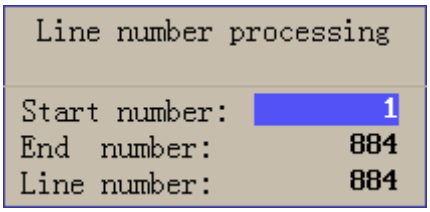
Select "English"-"Motion Controller", select ".mcp" file, and press " 

" to upgrade.

Chapter 3 Introduction to Advanced Functions

3.1 Line number processing

Press "Shift" + "Modify" key to select "Line Number Processing". After setting the processing parameters, press the "OK" key. At this time, you need to set the "start line", press the "Modify" key to set, and press the "OK" key after setting. Press "OK" again to process.

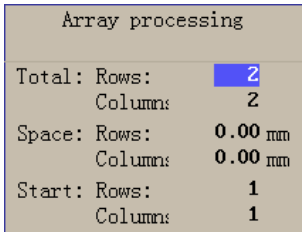


3.2 Breakpoint processing

Press "Shift" + "Modify" key to select "Breakpoint Processing". After setting the processing parameters, press the "OK" key.

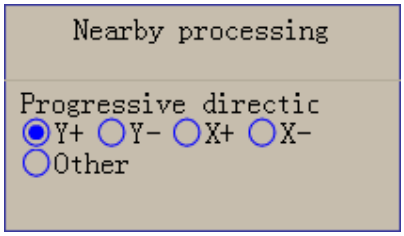
3.3 Array processing

Press "Shift" + "Modify" key to select "Array Machining". After setting the processing parameters, press the "OK" key. At this time, you need to set the relevant parameters, press the "Modify" key to set, and press the "OK" key after setting. Press "OK" again to process.



3.4 Nearby processing

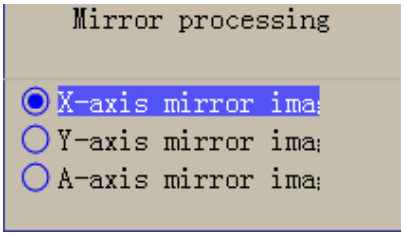
After moving to the position to be processed, press the "Shift" + "Modify" keys to select "Nearest point processing". After setting the processing parameters, press the "OK" key. At this time, you need to set the "progressive direction", press the "Modify" key to set, and press the "OK" key to start calculation and search after setting. After finding it, press "OK" to process.



Progressive direction: This parameter can significantly improve the calculation and search speed of nearby point processing. For relief carving, the overall direction of progression is consistent. If the progress direction is disordered, you can choose "other".

3.5 Mirror processing

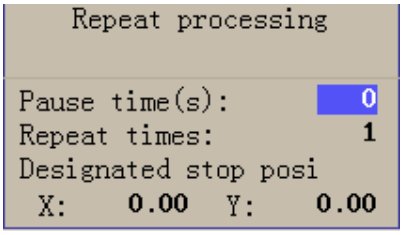
Press "Shift" + "Modify" key to select "Mirror Image Processing". After setting the processing parameters, press the "OK" key. At this time, you need to set the relevant parameters, press the "Modify" key to set, and press the "OK" key after setting. Press "OK" again to process.



3.6 Repeat processing

Press "Shift" + "Modify" key to select "Repeat Processing". After setting the processing parameters, press the "OK" key. At this time, you need to set the relevant parameters, press the "Modify" key to set,

and press the "(0,0) OK" key after setting. Press "(0,0) OK" again to process.



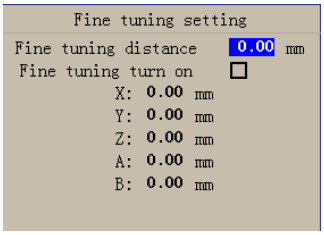
3.7 Save the workpiece origin

Press "Shift" + "修改 Modify" key to select "Save Workpiece Origin".

3.8 Read the workpiece origin

Press "Shift" + "修改 Modify" key to select "Read workpiece origin".

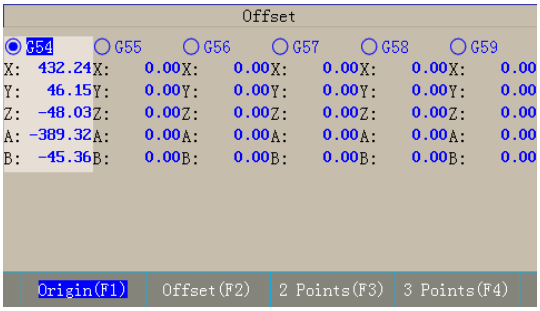
3.9 Online fine-tuning



Press the K key corresponding to "fine tuning" to fine-tune the parameter settings. After the fine-tuning function is turned on, during processing, press the "4 X-","6 X+", "2 Y+","8 Y-", "3 Z+","3 Z+"keys to adjust left and right, front and back, and depth.

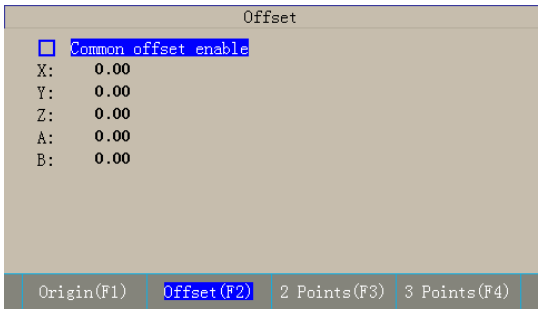
3.10 Set and select G54~G59 coordinate system

In the main interface, press the K key corresponding to "coordinate offset". Press "K1" key to select "Origin", press "修改 Modify" key to switch between G54~G59 or change the value, press "2 Y+" key and "8 Y-" key to select axis option, press "(0,0) OK" key to save and return to the main interface, press "ESC" key to return .

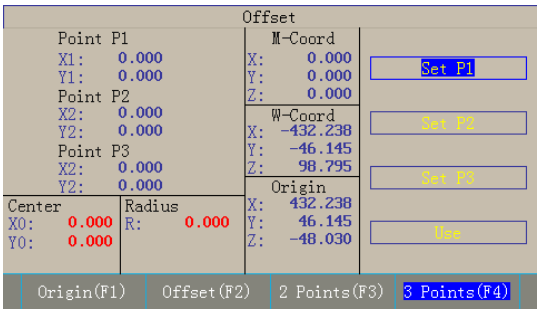


3.11 Public bias

In the main interface, press the K key corresponding to "coordinate offset". Press the "K2" key to select "Offset", press the "修改 Modify" key to modify the value or check whether the check is effective, press the "2 Y+" and "8 Y-" keys to move the cursor option, press the "(0,0) OK" key to save and return to the main interface, press the "ESC" key to return .



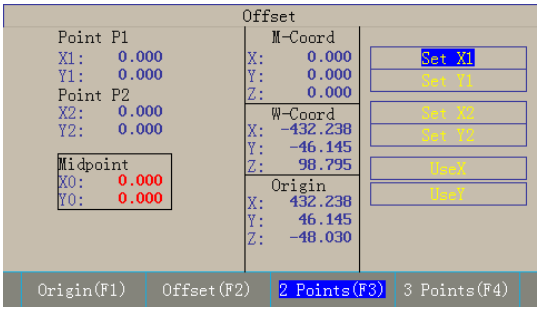
Invalid public bias



Public bias effective

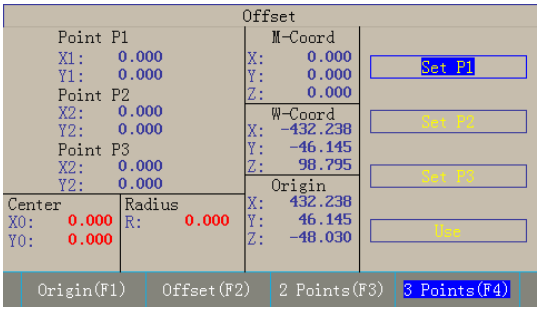
3.12 Calculate midpoint

In the main interface, press the K key corresponding to "Offset". Press the "K3" key to select the "2 Points", press the "(0,0) OK" key to execute, press the "2 Y+" and "8 Y-" keys to move the cursor, and press the "ESC" key to return.



3.13 Calculate the center of the circle

In the main interface, press the K key corresponding to "Offset". Press the "K4" key to select the "3 Points", press the "(0,0) OK" key to execute, press the "2" and "8" keys to move the cursor, and press the "" key to return.



Chapter 4 Introduction to Parameters

4.1 Machine parameters

After pressing the "  " key, then press the "  " key, enter the password "6666", and press "  ", you can see the "Machine Parameters Setting" option.

Machine parameter list

name	unit	Description	other
Port settings		Set input and output functions, port number and level	
Pulse equivalent	mm/pulse	The distance the axis moves for every 1 pulse	
Machine size	mm	The positive stroke is a positive value or 0, and the negative stroke is a negative value or 0 refers to the effective movement stroke of the machine tool. Set the maximum machining size of each axis. Please refer to the actual maximum processing range of the machine tool for setting.	
Soft limit status		After it is turned on, the mechanical coordinates of single-axis movement will not exceed the range of machine size setting. It is mainly a protective measure to prevent the machine tool from being damaged by wrong actions such as processing files exceeding the actual processing size of the machine tool and mechanical crash.	
Home direction		It is determined according to the mechanical zero position of the machine tool.	
Home speed	mm/min	Set the speed at which the machine tool returns to the mechanical zero point.	
Backward distance	mm	After the position of the zero point sensor is determined, the machine tool will retreat a certain distance and then set the last stopped position as the mechanical zero point position.	
Home limit dual purpose		After opening, the zero limit switch will also have a hard limit function.	
Prompt to return to		After opening, the main interface will pop up a	

zero when starting		window prompting to return to zero	
Do I have to zero when starting;		After opening, it cannot be processed without returning to zero	
A axis does not return to zero when all zeros are returned		After it is turned on, the A-axis will not automatically return to the zero point when all zeros are returned, unless the A-axis performs a single-axis zero return operation	This parameter is valid for HC-204A, and invalid for other models
Home encoder position		The number of pulses at the mechanical zero position recorded by the absolute encoder driver	
Default turn on		After opening, the processing file header will automatically start the spindle without M03	
Max analog voltage	V	Voltage value corresponding to maximum speed	The maximum voltage of this system can only be set to 10V
Maximum speed	rpm	After the spindle is turned on, the adjustment magnification will not exceed this value	
Minimum speed	rpm	After the spindle is turned on, the adjustment magnification will not be lower than this value	
Default speed	rpm	The default speed of the spindle	
Delay of on	ms	The time to wait for the spindle motor to start to the maximum speed when starting processing. The unit is milliseconds (ms).	
Delay of off	ms	The time to wait for the spindle motor to stop rotating when turning off the spindle. The unit is milliseconds (ms).	
Spindle no running check time	ms	For the inline tool magazine tool change system, the spindle speed must be zero during tool change. Before the spindle executes the tool change command, it will continuously check whether the spindle reaches zero speed during the spindle zero speed detection period. If the spindle zero speed detection period is exceeded, the drive has not returned the zero value signal, the system will automatically stop the tool change and end Processing.	HC-203C is effective
Analog compensation setup		Algorithm to make the output voltage value more accurate	Please see the relevant video for the setting method
Safe distance between spindle	mm	When the distance between the two spindles is lower than the safety distance, it will stop moving and prompt.	This parameter is valid for HC-205A, and invalid for

			other models
Motor direction		Adjust the rotation direction of the motor	
Manual high speed	mm/min	Manual high-speed movement speed in manual continuous mode	
Manual low speed	mm/min	Manual low-speed movement speed in manual continuous mode	
JOG high-speed distance	mm	High-speed jog distance in manual jog mode	
JOG low-speed distance	mm	Low speed jog distance in manual jog mode	
Max speed	mm/min	The maximum speed of each axis.	
Safety height	mm	Refers to the tool lifting height when the system executes actions such as returning to the workpiece origin and stopping. Note: The value here is relative to the workpiece origin, that is, if the user does not set the workpiece origin correctly before executing the above actions, the tool may still touch the workpiece, etc. When the set safety height is greater than the maximum height of the machine tool Z axis, the safety height will automatically be equal to the maximum height of the machine tool Z axis.	
Start speed	mm/min	The factory parameters of the motor generally include the take-off frequency parameter. But after the machine tool is assembled, the value may change, and it will generally decrease, especially when doing a load movement. Therefore, the setting parameters are best determined by actual measurement after referring to the factory parameters of the motor. The default is 120.	

Measurement mode		**0: Floating tool calibration:** Perform tool calibration at the current position, use the tool calibration thickness parameter, and the system automatically sets the Z axis workpiece coordinate. **1: Fixed tool calibration:** Perform tool calibration at a fixed position on the machine tool, and determine the position by setting the parameters of the tool calibration machine's mechanical coordinates. **2: Jade tool setting:** distinguish the first tool setting and the tool setting after changing	
Block thickness	mm	When the tool setting mode is set to floating tool setting, this interface will appear and allow changes. When using, please place the tool setter on the surface of the workpiece. When the tool setting is completed, the Z-axis workpiece origin coordinate = the mechanical coordinate when the tool setting signal is detected-the thickness of the tool setter, so the user must input the thickness of the tool setting block To the parameters.	
Sensor block position	mm	When the mechanical coordinates of the tool setter exceed the size of the machine tool, the mechanical coordinates of the tool setter will automatically be equal to the maximum allowable value.	
Z initial position	mm	The starting point of tool setting, from this position, the tool moves down slowly for tool setting. It is the mechanical coordinate value.	
Lowest position	mm	The lowering position limit of the tool during tool setting. It is the mechanical coordinate value.	
Measurement speed	mm/min	The speed when the tool tip is close to the tool setter.	
Measurement ation			
Measurement before processing			

Measurement after tool changer			
Tool magazine			HC-203A is invalid
Parameter setting of tool changer			HC-203A is invalid
Rolling rod setting		The system supports 2 rolling rods. During processing, when the Y-axis mechanical coordinate is between the rear position and the front position, the rolling rod will automatically press down. Y rear position <Y front position.	
Absolute encoding direction		Counting direction of absolute encoder drive	
Multi table function		After opening, there will be one more worktable stroke and can be switched. The workpiece origins of the two worktables are set and used separately, without mutual interference, suitable for three-axis + four-axis engraving models.	HC204A effective
Sub table size		Effective after the multi-workbench function is turned on	HC204A effective
Workpiece origin locking		After setting the workpiece origin correctly, tick the corresponding axis, the user can no longer set the workpiece origin of the corresponding axis, and the current workpiece origin is always maintained.	

4.2 Processing parameters

Name	Unit	Description	Other
Empty-way speed	mm/min	Generally, it refers to the movement speed when the cutting task is not executed, which is expressed by G00 command.	
Processing speed	mm/min	Generally, it refers to the movement speed when the cutting task is executed. The linear movement is expressed by G01 instruction.	
Turning speed	mm/min	It is used to describe the acceleration and deceleration capabilities of multiple feed axes when turning.	
Uniaxial acceleration	mm/s²	Used to describe the acceleration and deceleration capacity of a single feed axis. This index is determined by the physical characteristics of the machine tool, such as the quality of the moving part, the torque of the feed motor, the resistance, and the cutting load. The larger the value, the smaller the time spent in acceleration and deceleration during the movement and the higher the efficiency. Generally, for stepper motors, the value is between 300 and 800, and for servo motor systems, it can be set between 400 and 1200. In the setting process, start to set a little smaller, run for a period of time, repeat various typical exercises, pay attention to observation, if there is no abnormal situation, then gradually increase. If an abnormal situation is found, reduce the value and leave a 50% to 100% insurance margin.	
Processing Acc	mm/min	G01, G02, G03 command the acceleration of the feed rate. It is used to describe the acceleration and deceleration capacity of multiple feed axes in linkage. It determines the maximum speed of the machine tool in circular motion. The greater the value, the greater the maximum allowable speed of the machine tool during circular motion. The multi-axis linkage performance of the machine tool embodied by this index is a comprehensive index that is difficult to calculate	

		directly, but generally the larger the single-axis acceleration, the larger the value. Setting this value reasonably can improve the processing efficiency and reduce the vibration caused by the acceleration and deceleration of the machine tool during turning. But if the set value is too large, it will increase the vibration of the machine tool and even cause the motor to lose step. Generally, for a machine tool composed of a stepper motor system, the value is between 400 and 1000, and for a servo motor system, it can be set between 1000 and 5000. If it is a heavy machine tool, the value should be smaller. In the setting process, start to set a little smaller, run for a period of time, and repeat various typical linkage movements, pay attention to observation, if there is no abnormal situation, then gradually increase. If an abnormal situation is found, reduce the value and leave a 50% to 100% insurance margin. Non-positive numbers are not allowed to be set.	
Empty-way Acc	mm	G00 command acceleration of rapid traverse speed	
Processing acc of rotating shaft	mm/s^2		
Turning speed of rotating shaft	mm/s		
Speed ratio of rotating shaft		In 4-axis linkage, if the processing speed is unchanged, the larger the value, the faster the processing speed of the rotary axis.	HC-204A default value 2 other default values 1
Pause position setting		**0: Current position**: the machining is paused, the Z axis stops at the current position, and the spindle continues to open. **1: Specify Z-axis height**: When processing is paused, the Z axis rises by 20mm and the spindle is closed.	
Stop position setting		**0: Workpiece origin** When processing stops, the workpiece coordinate origin is returned, and the Z axis	

		rises to a safe height. **1: Current processing position** When processing is completed, stop at the current end point, and the Z axis will rise to a safe height. **2: Specify the XY axis position** Set the coordinate position of the XY axis, you can stop at the set coordinate position when the processing stops, and the Z axis will rise to a safe height. **3: Specify the XYZ axis position** Set the coordinate position of the XYZ axis, and the machine can stop according to the set coordinate position when the processing stops.	
Stop position coordinate type		The coordinate type of the stop position for normal processing of the workpiece and repeated processing of the workpiece.	
Speed ratio control setting		○ Only control“G01”——The speed override adjustment during processing can only control the G0 speed ● Control overall speed —— The speed override during processing is adjusted to control all speeds of the machine tool	
Motion smoothing time	ms	The longer the motion smoothing time is, the smoother the processing effect will be. If the set value is too large, the right angle will become arc.	
Z-down speed	mm/min	In the cutting process of G01, in order to protect the tool, it is necessary to use the set feed speed when cutting downwards. The system automatically adopts the feed speed when the tool drops vertically by default.	
Falling mode		N1 G01 Z-1 N2 G01 X10 Y10 Z-2 1.Neither N1 nor N2 will adopt the Z feed speed; 2.N1 will adopt the Z feed speed, but N2 will not adopt the Z feed speed; 3.Both N1 and N2 will adopt the Z feed speed	
Reference length of short line segment	mm	Speed limit on G01 line segment less than the reference length	

Short line segments effective speed limit		Speed limit on G01 line segment less than the reference length	
Minimum number of subdivisions			
Chord error	mm	The smaller the value, the finer the arc machining of G02G03.	
Reference circle radius	mm/min	$\text{Arc speed} = \text{DefCircleSpeed} \times \sqrt{\frac{\text{Arc radius}}{\text{DefCircleRadius}}}$	
Reference circle speed	mm	$\text{Arc speed} = \text{DefCircleSpeed} \times \sqrt{\frac{\text{Arc radius}}{\text{DefCircleRadius}}}$	
Arc speed limit effective	mm	Limit the speed of G02 G03 arc command.	
G00 Motion type		**XY Plane and Z-Dir Motion**: In order to protect the workpiece, the G00 command motion is divided into XY plane motion and Z direction motion. If the Z axis is lower, then it will move in the XY plane first, then the Z axis, otherwise, first the Z axis and then the XY plane **Three axis linkage**: XYZ axes Moves Simultaneously.	
G00 down slow length	mm	When G00 drops the knife, the distance greater than the deceleration distance is the idle speed, and the distance not greater than the deceleration distance is the approach speed.	
G00 down slow speed	mm/min	When G00 drops the knife, the distance greater than the deceleration distance is the idle speed, and the distance not greater than the deceleration distance is the approach speed.	
Automatic opening rolling rod		When it is turned on, the rolling rod will be automatically raised and lowered according to the front position and the rear position during processing.	

4.3 System parameters settting

Name	Unit	Description	Other
F&S parameter configuration		Choose whether to execute the processing speed or spindle speed in the file.	
Sound settings		Choose whether to turn on button sound and alarm sound.	

Chapter 5 Introduction to Programming Instructions

5.1 G code command

Item	Function name	Remark
G00	Straight line fast positioning	
G01	Linear interpolation	
G02	Circular interpolation (clockwise)	
G03	Circular interpolation (counterclockwise)	
G04	Pause for specified time	
G17	Set X-Y work plane	
G18	Set Z-X work plane	
G19	Set Y-Z work plane	
G28	Reference return	
G43.4	Tool Point Follow (RTCP)	Four-axis swing head
G49	Tool point follow cancel	
G50.1	Mirror function cancel	
G51.1	Mirror function	
G53	Machine coordinate setting	
G54~G59	Workpiece coordinate setting	
G90	Absolute coordinate input method	
G91	Relative coordinate input method	
G500 X_	*Turn on the T1 and T2 mirroring functions. X represents the distance between the center line of the mirror image and the origin of the T1 workpiece*	Only valid for HC-205A
G501	Turn off the T1 and T2 mirroring function	Only valid for HC-205A

5.2 M code instruction

Item	Function name	Remark
M03	Turn on the spindle	
M04	Turn on the spindle	
M05	Turn off the spindle	
M08	Turn on the coolant	
M09	Turn off the coolant	
M12	Turn on the vacuum pump	
M13	Turn off the vacuum pump	
M30	End of program	

M34	Brush (dust hood) down	
M35	Brush (dust hood) up	
M60	Turn on the left positioning cylinder	
M61	Turn off the left positioning cylinder	
M62	Turn on the front positioning cylinder	
M63	Turn off the front positioning cylinder	
M64	Turn on right positioning cylinder	
M65	Turn off right positioning cylinder	
M66	Turn on rear positioning cylinder	
M67	Turn off rear positioning cylinder	
M68	Turn on side positioning cylinder	
M69	Turn off side positioning cylinder	
M99	Infinite loop processing	
M201	No.1 rolling rod descends	
M202	No.1 rolling bar rises	
M203	No.2 rolling rod descends	
M204	No.2 rolling bar rises	
M205	No.1 and No.2 rolling rods descend at the same time	
M206	No.1 and No.2 rolling rods rise at the same time	
M207	Turn on the automatic lifting function of the rolling rod	M201-M206 are invalid at this time
M208	Turn off the automatic lifting function of the rolling rod	
M330	T2 uses T1 workpiece origin processing	Only valid for HC-205A
M331	T2 uses T2 workpiece origin processing	Only valid for HC-205A
M324 L_	Set the distance between T1 and T2	Only valid for HC-205A

Appendix

一、Introduction of each model

Model	Description	Remark
HC-203A	Single-head 3-axis handheld control system	
HC-203B	Multi-head 3-axis handheld control system	
HC-203C	Single-head 3-axis inline tool change handheld control system	
HC-204A	Single-head 4-axis handheld control system	
HC-204R	Single pendulum head handheld control system with RTCP	
HC-205A	XZ dual-channel handheld control system	
HC-206A	Z4 channel handheld control system	

二、Introduction to the function of multi-head floating tool setting block

Set "Measure in unfixed position" in “Menu”-“Machine parameter setting”-“Tool measurement”-“Measurement mode”.

After installing the tool, press "Shift" + "K1","Shift"+"K2", "Shift" + "K3","Shift"+"K4"to directly set the tool lengths of T1, T2, T3 and T4. Press the "  " key to perform tool calibration to set the Z-axis workpiece origin. After tool calibration is completed, press the "  " key to process.

After the tool is broken, the tool length of the new tool needs to be reset.

三、HC-205A notes

1、Manual movement, tool measurement, single-axis zero return, and workpiece origin setting.

HC-205A has two modes in terms of control, T1 mode and T2 mode and T3 mode, which can be switched by the "K1","K2"and"K3"keys. The default left main axis is T1.

“  ”,“  ”, “  ”, “  ”, “  ”, “  ”are manual movements. In T1 mode and T2 mode, they control the movement of X1, Y, Z1 and X2, Y, Z2 axes, and in T3 mode, X1 and X2 Move manually at the same time, Z1 and Z2 move manually at the same time.

The "  " key measures T1 and T2 in T1 mode and T2 mode respectively.

“  ”+“  ”, “  ”+“  ”, “  ”+“  ” are single-axis zero return. In T1 mode and T2 mode, it controls the zero return of X1, Y, Z1 and X2, Y, Z2 axes respectively.

“  ”, “  ”, “  ” are to set the workpiece origin. In T1 mode and T2 mode, set the workpiece origin of X1, Y, Z1 and X2, Y, Z2 respectively. In T3 mode, set the workpiece origin of "X1, X2" and "Z1, Z2" at the same time.

2、X1 and X2 stroke settings

After X1 and X2 return to zero correctly, measure the distance between the tool noses on No.1 and No.2 spindles. This value is used as the positive travel of X1 (positive value) and the negative travel of X2 (negative value).

3、HC-205A tool offset setting

HC-205A sets the offset of the two spindles based on T1. Specific steps are as follows:

1. After successfully returning to zero on the main interface, manually move T1 to a suitable position and write down the mechanical coordinates of T1 (X1, Y1, Z1); press the "  " key to switch to T2 movement mode, and manually move T2 to the original T1 Position, write down the mechanical coordinates of T2 (X2, Y2, Z2);

2. Enter the “menu” - “machine parameter settings” - “tool settings” - “tool magazine” (you can also enter through the key combination "  " + "  "), and then press the"  "key, you can see that the upper left corner becomes CT2, and then directly set the T2 Offset, set the value of (X2-X1, Y2-Y1) to CT2's X and Y, and press OK to exit after setting.

4、Set tool length

Press "  " + "  " key, "  " + "  " key to directly set the tool length.

No need to actively set tool length in "Measure in fixed position" mode.

5、Safe distance between spindles

In the menu -> machine parameter setting -> spindle setting -> safe distance between spindles, if the distance between the two spindles is less than this value during processing, processing will stop.

6、Processing function

T1 single-head processing, T2 single-head processing, T1、T2 rotation processing, T1、T2 linkage processing, T1、T2 mirror processing.