

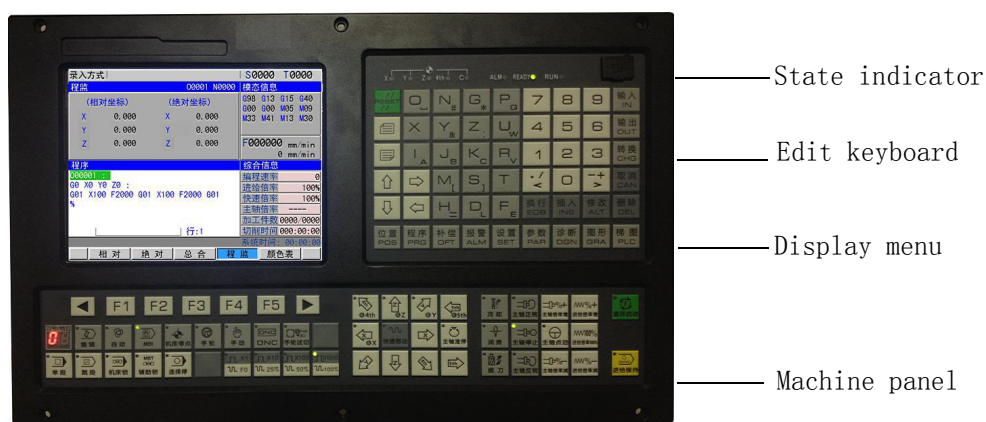
Volume II

Operation

1 Operator Panel

■ Panel layout

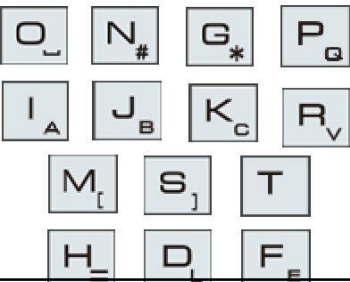
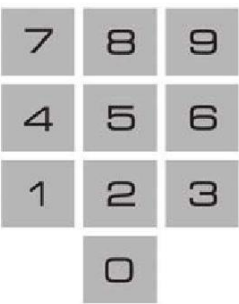
The GH1000MC machine center has an integrated operator panel. The layout of it is shown as following:



■ Status indication

	Axis zero return completion indicator
	Three color indicator

■ Edit Keypad

Button	Name	Function
	Reset key	CNC reset, feed and output stop, etc.
	Address keys	Input the address
		Double-address keys, switch between two addresses through pressing the buttons repeatedly
	Symbol keys	Three-address key, switch between three addresses through pressing the buttons repeatedly
	Digit keys	Input with the digits
	Input key	Confirm the data input, like the parameters and the compensation amount, etc.
	Output key	Start the communication output
	Change key	Change between the information and the display
	Edit keys	Inserting, rewriting and deleting the program and the field, etc during editing,

Button	Name	Function
	EOB key	For block end sign input
   	Cursor moving keys	For cursor moving control
 	Page keys	Page switching in a same interface

■ Display Menu

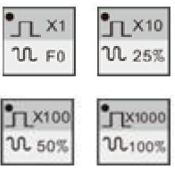
Menu key	Remark
	To enter position interface. There are RELATIVE POS, ABSOLUTE POS, INTEGRATED POS, POS&PRG pages in this interface.
	To enter program interface. There are PRG CONTENT, PRG STATE, PRG LIST, FILE DIRECTORY, 4 pages in this interface.
	To enter TOOL OFFSET interface. There are TOOL OFFSET, MARRO variables. OFFSET interface displays offset values; MARRO for CNC macro variables.
	To enter alarm interface. There are CNC, PLC ALARM and ALARM Log pages in this interface.
	To enter Setting interface. There are SWITCH, PASSWORD SETTING, DATE & TIME, SETTING (G54~G59), GRAGH SET and TRACK pages in this interface.
	To enter BIT PARAMETER, DATA PARAMETER, PITCH COMP interfaces (switching between each interface by pressing repeatedly).
	To enter DIAGNOSIS interface. There are CNC DIAGNOSIS, PLC STATE, PLC VALUE, VERSION MESSAGE interfaces (switching between each interfaces by pressing the key repeatedly). CNC DIAGNOSIS, PLC STATE, PLC VALUE interfaces display CNC internal signal state, PLC addresses, data state message; the VERSION MESSAGE interface displays CNC software, hardware and PLC version No.
	To enter Ladder interface. There are PLC information, PLC ladder, PLC parameter PLC diagnosis interfaces (switching between each interfaces by pressing the key repeatedly).

■ Machine Panel

The keys function in GH1000MC machine panel is defined by PLC program (ladder), see their function significance in the machine builder's manual.

The functions of the machine panel keys defined by standard PLC program are as follows:

Key	Name	Function explanation	Function mode
	Feed Hold key	Dwell commanded by program, MDI	Auto mode, DNC, MDI mode
	Cycle Start key	Cycle start commanded by program, MDI	Auto mode, DNC, MDI mode
	Feedrate Override keys	For adjustment of the feedrate	Auto mode, DNC, MDI mode, Edit mode, Machine zero mode, MPG mode, Single Step mode, MANUAL mode
	Rapid override keys	For adjustment of rapid traverse	Auto mode, DNC, MDI mode, Machine zero mode, MANUAL mode
	Spindle override keys	For spindle speed adjustment (spindle analog control valid)	Auto mode, DNC, MDI mode, edit mode, Machine zero mode, MPG mode, Step mode, MANUAL mode
	JOG key	For spindle Jog ON/OFF	Machine zero mode, MPG mode, Single Step mode, MANUAL mode
	Lubricating key	For machine lubrication ON/OFF	Machine zero mode, MPG mode, Single Step mode, MANUAL mode
	Cooling key	For coolant ON/OFF	Auto mode, MDI mode, Edit mode, Machine zero mode, MPG mode, Step mode, MANUAL mode
	Spindle control keys	Spindle CW Spindle stop Spindle CCW	Machine zero mode, MPG mode, Single Step mode, MANUAL mode
	Rapid traverse key	For rapid traverse /feedrate switching	Auto mode, DNC, MDI mode, Machine zero mode, MANUAL mode

Key	Name	Function explanation	Function mode
 	X axis feed key	For positive/negative moving of X, Y, Z axis in Manual, Step mode	Machine zero mode, Step mode, MANUAL mode
 	Y axis feed key		
 	Z axis feed key		
 	4th axis feed key		
	MPG/Step increment and Rapid override selection key	Move amount per handwheel scale 0.001/0.01/0.1 mm Move amount per step 0.001/0.01/0.1 mm Rapid override F0, F25%, F50%, F100%	Auto mode, MDI mode, Machine zero mode, MPG mode, Step mode, MANUAL mode
	Optional stop	Execute M01 stop when optional stop is valid	Auto mode, DNC, MDI mode
	Single Block key	For switching of block/blocks execution, Single block lamp lights up if Single mode is valid	Auto mode, DNC, MDI mode
	Block Skip key	For skipping of block headed with "/" sign, if its switch is set for ON, the Block Skip indicator lights up	Auto mode, DNC, MDI mode
	Machine Lock key	If the machine is locked, its lamp lights up, and X, Z axis output is invalid.	Auto mode, DNC, MDI mode, Edit mode, Machine zero mode, MPG mode, Step mode, MANUAL mode
	M.S.T. Lock key	If the miscellaneous is function locked, its lamp lights up and M, S, T function output is invalid.	Auto mode, DNC, MDI mode
	Edit mode key	To enter Edit mode	Auto mode, DNC, MDI mode, Machine zero mode, MPG mode, Step mode, MANUAL

Key	Name	Function explanation	Operation mode
	Edit mode key	To enter Edit mode	Auto, MDI, Machine zero return, Manual, Step, MPG, Program zero return
	Auto mode key	To enter Auto mode	MDI, Edit, Machine zero return, Manual, Step, MPG, Program zero return
	MDI mode key	To enter MDI mode	Auto, Edit, Machine zero return, Manual, Step, MPG, Program zero return
	Machine zero return mode key	To enter Machine zero return mode	Auto, MDI, Edit, Manual, Step, MPG, Program zero return
	Step/MPG mode key	To enter Step or MPG mode (one mode by parameter)	Auto, MDI, Edit, Machine zero return, Manual, Program zero return
	Manual mode key	To enter Manual mode	Auto, MDI, Edit, Machine zero return, Step, MPG, Program zero return
	DNC mode key	To enter DNC mode	Auto, MDI, Edit, Machine zero return, Step, MPG, Manual

■ Overview of the Operation Modes

There are 7 modes in GH1000MC, which are Edit, Auto, MDI, Machine zero, Step/MPG, Manual, DNC.

■ Edit mode

In Auto mode, the operation of part program setup, deletion and alteration can be performed.

■ Auto mode

In Auto mode, the program is automatically run.

■ MDI mode

In MDI mode, the operation of parameter input, command blocks input and execution can be performed.

■ Machine zero return mode

In machine zero return mode, the feeding axis returning to the machine zero can be operated respectively.

■ MPG / Step mode

In MPG / Step mode, the moving is performed by an increment selected by CNC system.

└. Manual mode

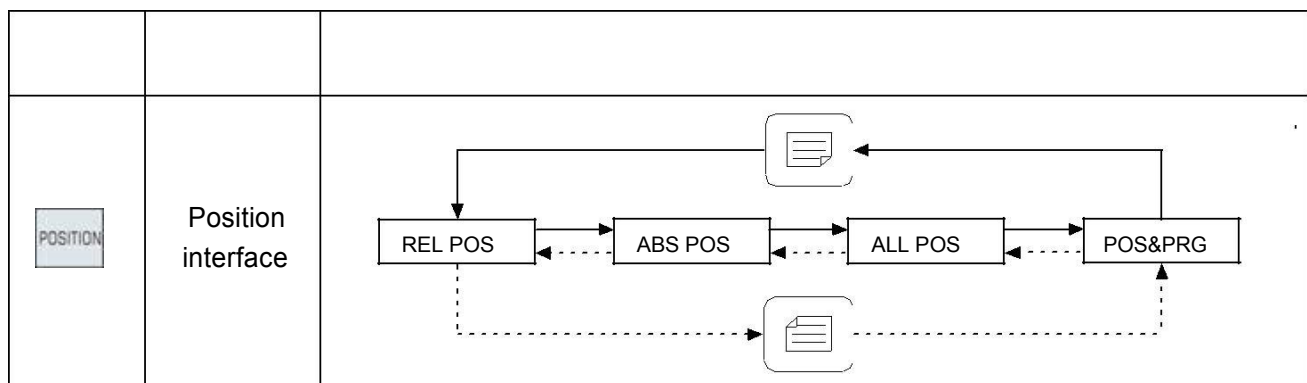
In Manual mode, the manual feeding, manual rapid rate, the feedrate override adjustment, the rapid override adjustment, the spindle start/stop, the coolant switch, the lubricant switch, the spindle inch and the manual tool change, etc. can be executed.

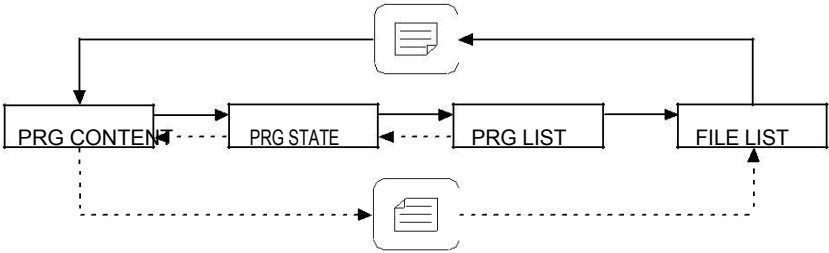
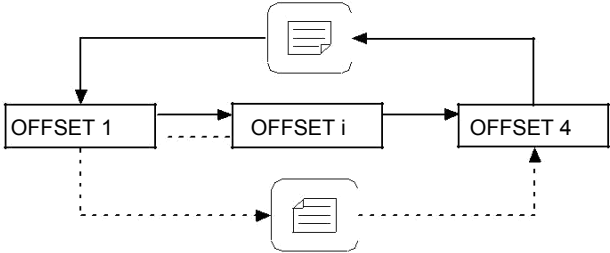
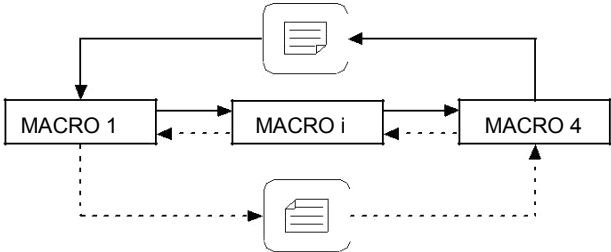
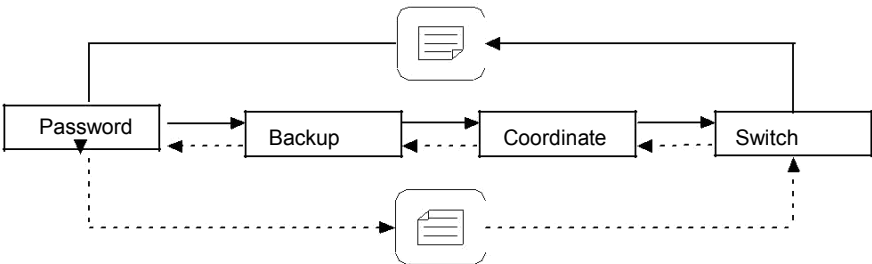
■ DNC mode

In DNC mode, the program is run by DNC mode.

■ Display Interface

There are 9 interfaces such as Position, Program etc., and there are multiple pages in each interface. Each interface (page) is separated with the operation mode. See the following figures for the display menu, display interface and page layers:



PROGRAM	Program interface	 <pre>graph LR; PC[PRG CONTENT] --> PS[PRG STATE]; PS --> PL[PRG LIST]; PL --> FL[FILE LIST]; FL --> PC; PC -.-> D1(()); D1 -.-> D2(()); D2 -.-> FL; PS -.-> D1; PL -.-> D2</pre>
OFFSET	Tool offset interface	 <pre>graph LR; O1[OFFSET 1] --> Oi[OFFSET i]; Oi --> O4[OFFSET 4]; O4 --> O1; O1 -.-> D1(()); D1 -.-> D2(()); D2 -.-> O4; Oi -.-> D1; O4 -.-> D2</pre>
		 <pre>graph LR; M1[MACRO 1] --> Mi[MACRO i]; Mi --> M4[MACRO 4]; M4 --> M1; M1 -.-> D1(()); D1 -.-> D2(()); D2 -.-> M4; Mi -.-> D1; M4 -.-> D2</pre>
ALARM	ALARM interface	Alarm information
	ALARM log interface	Alarm log
SETTING		 <pre>graph LR; P[Password] --> B[Backup]; B --> C[Coordinate]; C --> S[Switch]; S --> P; P -.-> D1(()); D1 -.-> D2(()); D2 -.-> S; B -.-> D1; C -.-> D2</pre>

PARAMETER	Bit parameter	The diagram illustrates the relationships between different parameter types: Bit parameter: A sequence of boxes labeled BITPAR 1, BITPAR n, BITPAR 1, BITPAR i, and BITPAR n. Solid arrows connect BITPAR 1 to BITPAR n, BITPAR n to BITPAR 1, BITPAR 1 to BITPAR i, and BITPAR i to BITPAR n. A document icon is positioned above the first BITPAR 1 box. Data parameter: A sequence of boxes labeled BITPAR 1, BITPAR n, BITPAR 1, BITPAR i, and BITPAR n. Dashed arrows connect BITPAR 1 to BITPAR n, BITPAR n to BITPAR 1, BITPAR 1 to BITPAR i, and BITPAR i to BITPAR n. A document icon is positioned above the first BITPAR 1 box. Screw-pitch parameter: A sequence of boxes labeled Screw-pitch 1, Screw-pitch 2, and Screw-pitch n. Solid arrows connect Screw-pitch 1 to Screw-pitch 2, Screw-pitch 2 to Screw-pitch n, and Screw-pitch 1 to Screw-pitch n. A document icon is positioned above the Screw-pitch 1 box.
	Data parameter	
	Screw-pitch parameter	
DIAGNOSE	diagnosis CNC	
	PLC Signal	
	Machine tool Softpanel	Machine Soft panel
	Help	
	Edition Information	Edition Information
PLC	PLC Information	
	PLC Ladder	
	PLC parameter	
	PLC diagnosis	

■ Position interface

Press  to enter Position interface, which has four interfaces such as ABSOLUTE POS,

RELATIVE POS, INTEGRATED POS and POS&PRG, and they can be viewed by  or  keys. 1) ABSOLUTE POS **display** interface

The X, Y, Z coordinates displayed are the absolute position of the tool in current workpiece coordinate system, as CNC power on, these coordinates are held on and the workpiece coordinate system is specified by G92、G54-G59.



Note It displays “PRG. F” in Auto, MDI mode; “JOG. F” in Machine zero, Manual mode; “HNDL INC” in MPG mode; “STEP INC” in Step mode.

ACT. F: actual speed after feedrate override in a machining.

FED OVRI: an override by feedrate override switch.

G CODE: modal value of 01 group G code and 03 group G code

PART CNT: part number plusing 1 when M30(or M99 in the main program) is executed

CUT TIME: Time counting starts if Auto run starts, time units are hour, minute and second

RAPID OVRI: It displays current rapid rapid override

SPINDLE OVRI: It displays spindle override when BIT4 of NO.001 was set 0

S0000: Feedback spindle speed of spindle encoder, and spindle encoder is a must

Current tool : Tool No. specified by T command in program

Tool offset: H00 tool length compensation of current processing program;

D00 radius compensation of current processing program

The parts counting and the cut time are memorized at power-down and the clearing ways for them are as following:

PART CNT clearing: press  key then press  key.

CUT TIME clearing: press  key then press  key.

2) RELATIVE POS display page

The X, Y, Z axis coordinates displayed are the current position relative to the relative reference point, and they are held on at CNC power on. They can be cleared at any time. If X, Y, Z axis relative coordinates are cleared, the current position will be the relative reference point.

The clearing steps of X, Y, Z axis relative coordinates:

In RELATIVE POS page, press and hold  key till the “X” in the page blinks, press  key to clear X coordinate;

In RELATIVE POS page, press and hold  key till the “Y” in the page blinks, press  key to clear Y coordinate;

In RELATIVE POS page, press and hold  key till the “Z” in the page blinks, press  key to clear Z coordinate;



3) INTEGRATED POS display page

In INTEGRATED POS page, the RELATIVE, ABSOLUTE, MACHINE coordinate, DIST TO GO (only in Auto and MDI mode) are displayed together.

The displayed value of MACHINE coordinate is the current position in the machine coordinate system which is set up according to the machine zero.

DIST TO GO is the difference of the target position by block or MDI command to the current position.

The display page is as following:



■ Program page set



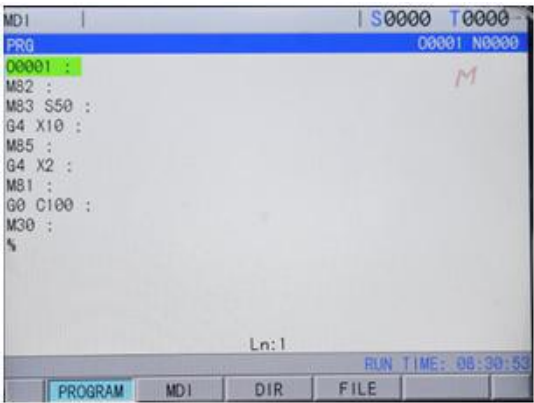
Press  to enter Program interface, which has four pages such as PRG CONTENT, PRG STATE, PRG LIST and FILE CONTENT in non-Edit modes, and they can be viewed by pressing key  repeatedly or pressing the corresponding software key.

1) Program content page

In the program content page, the content of the current program is displayed. In Edit mode,

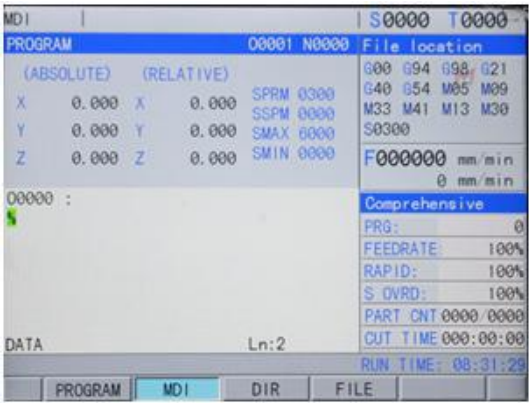


press  and  keys to check the program content forward or backward.



2) Program state

In the program content page, Press  Key and enter the program state page



3) Program contents page



In the program status page , press key  and enter the program contents page. At this interface, all of the processing procedures are listed.

The contents of the program Files page

(a) Existing program number:

Shows the number of programs that have been stored in the CNC (including subroutine)

(b) Remaining number of program:

Shows the number of programs that can be stored in CNC

(c) Storage capacity:

The amount of memory that the CNC has been stored in (KB)

(d) Residual memory:

The remaining capacity of the CNC storage part program(KB)

(e)Program contents :

The program number of the part program is displayed in the name of the part program.

(f) Program size:

The size of the storage space of the CNC program



4) File contents page

In the program contents page, press CHANGE enter the file contents page. Page display is as follows:

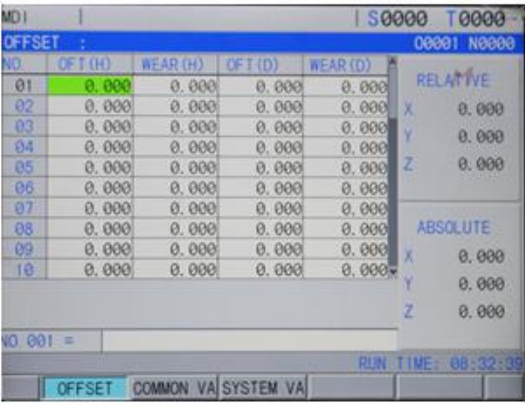


■ **Tool offset, macro variable and tool life management interface**

 is a compound key, press  key once in other page, it enters the TOOL OFFSET page, press  key again, it enters the MACRO interface.

1) OFFSET interface

There are 4 tool offset pages in this interface, and 32 offset No. (No.001~No.032) available for user, which can be shown as following by pressing  or  keys.



NO.	OFF (H)	WEAR (H)	OFF (D)	WEAR (D)
01	0.000	0.000	0.000	0.000
02	0.000	0.000	0.000	0.000
03	0.000	0.000	0.000	0.000
04	0.000	0.000	0.000	0.000
05	0.000	0.000	0.000	0.000
06	0.000	0.000	0.000	0.000
07	0.000	0.000	0.000	0.000
08	0.000	0.000	0.000	0.000
09	0.000	0.000	0.000	0.000
10	0.000	0.000	0.000	0.000

NO.001 =
 RUN TIME: 08:32:39

2. COMMON VARIABLE interface

There are 20 windows in this interface, which can be shown by pressing  or  key. In Macro window there are 600 (No.100 ~ No.199 and No.500 ~ No.999) macro variables which can be specified by macro command or set by keypad.



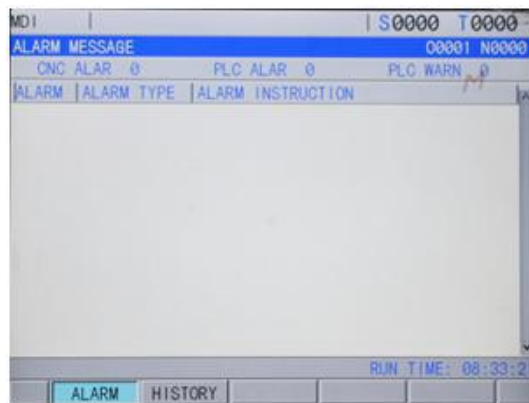
NO.	DATA	NO.	DATA	NO.	DATA
100		110		120	
101		111		121	
102		112		122	
103		113		123	
104		114		124	
105		115		125	
106		116		126	
107		117		127	
108		118		128	
109		119		129	

COMMON VARIABLES
NO.101 =
 RUN TIME: 08:32:54

■ ALARM interface

1. Alarm:

Press  key to enter alarm interface, which can be viewed by  or  key, the window is as follows:

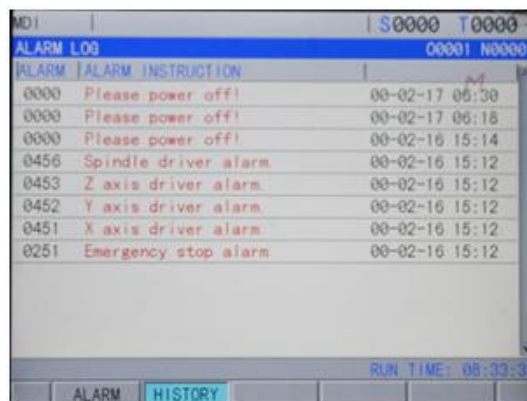


Note: Alarm clearing: It may clear alarms by pressing  (it should press  and  keys together to clear No.100 alarm). If the current alarm window is as follows:

2) Alarm log:

Press  key again to enter WARN LOG interface, 200 messages can be viewed by pressing  or  key.

Sequence of warn log: The latest log message is shown on the forefront of the 1st window, and the others queue in sequence. If the messages are over 200, the last one will be cleared.



Note: Manual clear alarm log: press  +  in 2-level password to clear all log message.

■ Setting interface



is a compound key, press  key in other window, it enters the SETTING interface,

press it again, it enters the GRAPHIC interface. Press key  repeatedly, it switches between SETTING and GRAPHIC interfaces.

1. SETTING interface

There are 5 windows in this interface, which can be viewed by  or  key.

SWITCH SETTING: It is used for the parameter, program, auto sequence No. on-off state.

PARM SWT: when it is turned for ON, the parameters are allowed to be altered; it is turned for OFF, the parameters are forbidden to be altered.

PROG SWT: when it is turned for ON, the programs are allowed to be edited; it is turned for OFF, the programs are forbidden to be edited.

AUTO SEG: when it is turned for ON, the block No. is created automatically; it is turned for OFF, the block No. is not created automatically, but manually if needed.



DATA OPERATION: In this window, the CNC data (such as bit parameter, data parameter, screw-pitch parameter, tool offset) can be backup and restored.

Restore original value:Restore parameters,tool compensation,screw-pitch to system default

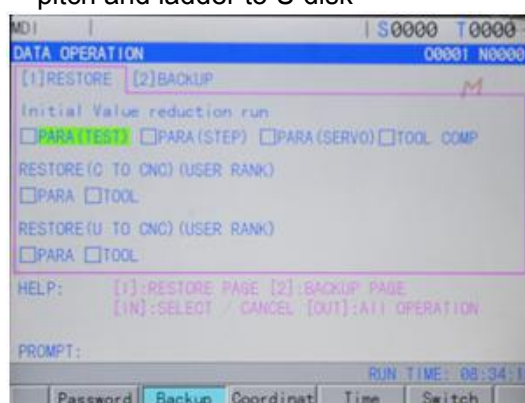
value. C disk data restore to CNC:Restore data files which are backup to system disk to system

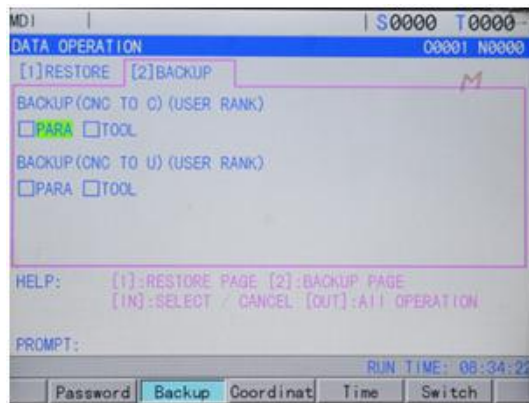
U disk data restore to CNC:Restore data files which are backup to U disk to system

CNC data backup to C disk:Backup current parameters,tool compensation,screw-pitch

and ladder to system disk

CNC data backup to U disk :Backup current parameters,tool compensation,screw-pitch and ladder to U disk





PASSWORD SETTING: For user operation level display and setting

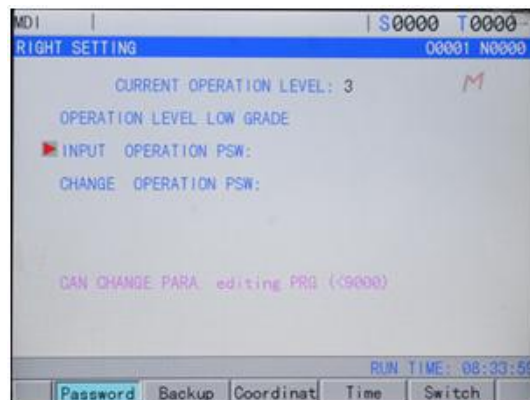
By descending sequence the password of GH1000MC is classified for 4 levels, which are machine builder (2) level, equipment management (3) level, technician (4) level, machining operation (5) level.

Machine builder level: The CNC bit parameter, data parameter, screw-pitch parameter, tool offset data, part program edit(including macro), PLC ladder editing and alteration, ladder upload and download operations are allowed;

Equipment management level: Initial password is 12345, the CNC bit parameter, data parameter, tool offset data, part program edit operations are allowed;

Technician level: Initial password is 1234, tool offset data (for toolsetting), macro variables, part program edit operations are allowed; but the CNC bit parameter, data parameter, screw-pitch parameter operations are forbidden.

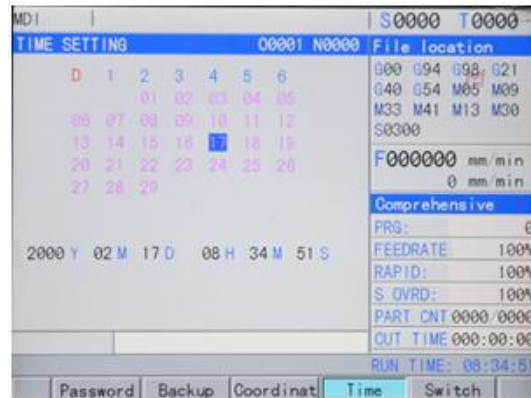
Operation level: No password. Only the machine panel operation is allowed, the operations of part program edit and selection, the alteration operations of CNC bit parameter, data parameter, screw-pitch parameter, tool offset data are forbidden.



Time, data: display current time and date.

Press  to enter the change mode, press  ,  to change Year/ Month/ Day/ Minute/

Second, press  ,  to change the value. If the system escapes the mode,  is pressed again.



1.3.6 BIT PARAMETER, DATA PARAMETER, SCREW-PITCH COMP interfaces



is a compound key, it enters BIT PARAMETER, DATA PARAMETER and SCREW-PITCH COMP interfaces by pressing this key repeatedly.

1) BIT PARAMETER interface



Press  key, it enters BIT PARAMETER interface, there are 60 bit parameters which are displayed by 2 windows in this interface, and they can be viewed or altered by pressing  or  key to enter the corresponding window. It is shown as follows:

As is shown in this window, there are 2 parameter rows at the window bottom, the 1st row shows the meaning of a bit of a parameter where the cursor locates, the bit to be displayed can be

positioned by pressing  or  key. The 2nd row shows the abbreviation of all the bits of a parameter where the cursor locates.

MDI		S0000 T0000	
STATUS PARAMETER		00001 N0000	
NO.	DATA	NO.	DATA
001	00001000	011	00000000
002	00000000	012	01110001
003	00010100	013	11000001
004	00000000	014	00000000
005	00011000	015	00000000
006	00001001	016	10110010
007	00000000	017	00000101
008	00001101	018	00100000
009	10011111	019	11010100
010	00001000	020	00000100
021	00000111	022	10000001
023	00000000	024	00000000
025	00000000	026	00000000
027	00000000	028	00000000
029	00110000	030	00110000
030	00000000		
**** SPTY SOHW **** (N)			
BIT0: (0: In mm 1: In inches)/input			
NO.001 =			
RUN TIME: 08:35:19			
BITPAR		NUMPAR	
PITCH COM		USER PARA	

2) DATA PARAMETER interface

Press  key repeatedly ( key if in BIT PARAMETER interface), it enters DATA

PARAMETER interface, and they can be viewed or altered by pressing or  or  to enter the corresponding window. It is shown below:

As is shown in this window, there is a cue line at the window bottom, it displays the meaning of the parameter where the cursor locates.

MDI		S0000 T0000	
NUM PARAMETER		00001 N0000	
NO.	DATA	NO.	DATA
000	1	010	-9999.0000
001	1	011	9999.0000
002	1	012	-9999.0000
003	1	013	9999.0000
004	1	014	-9999.0000
005	1	015	9999.0000
006	1	016	-9999.0000
007	1	017	9999.0000
008	1	018	-9999.0000
009	1	019	9999.0000
020		020	0.0000
021		021	0.0000
022		022	0.0000
023		023	0.0000
024		024	0.0000
025		025	0.0000
026		026	0.0000
027		027	0.0000
028		028	0.0000
029		029	0.0000
MIN 1 MAX 65536			
A multiplication coefficient (CMR)			
NO.000 =			
RUN TIME: 08:35:29			
BITPAR		NUMPAR	
PITCH COM		USER PARA	

3) SCREW-PITCH COMP interface

Press  key repeatedly, it enters SCREW-PITCH COMP interface, there are 256 screw-pitch parameters which are displayed by 11 windows in this interface, and they can be viewed by pressing  or  key.

MDI		S0000 T0000	
PITCH COMP		00001 N0000	
NO.	X	Y	Z
000	0	0	0
001	0	0	0
002	0	0	0
003	0	0	0
004	0	0	0
005	0	0	0
006	0	0	0
007	0	0	0
008	0	0	0
009	0	0	0
010	0	0	0
011	0	0	0
012	0	0	0
013	0	0	0
014	0	0	0
015	0	0	0
016	0	0	0
017	0	0	0
018	0	0	0
019	0	0	0
020	0	0	0
021	0	0	0
022	0	0	0
023	0	0	0
NO.000			
RUN TIME: 08:35:33			
BITPAR		NUMPAR	
PITCH COM		USER PARA	

1.3.7 CNC DIAGNOSIS, PLC STATE, MACHINE SOFT PANEL, VERSION MESSAGE, HELP MESSAGE interfaces



is a compound key, it enters CNC DIAGNOSIS, PLC STATE, PLC VALUE, TOOL PANEL, VERSION MESSAGE interfaces by pressing this key repeatedly.

1) CNC DIAGNOSIS interface

The input/output signal state between CNC and machine, the transmission signal state between CNC and PLC, PLC internal data and CNC internal state can all be displayed via diagnosis. Press



key it enters CNC DIAGNOSIS interface, the keypad diagnosis, state diagnosis and miscellaneous function parameters etc. can be shown in this interface, which can be viewed by

pressing or key.

In CNC DIAGNOSIS window, there are two diagnosis rows at the window bottom, the 2nd row shows the meaning of a bit diagnosis No. where the cursor locates, the bit to be displayed can be positioned

by pressing or key. The 1st row shows the abbreviation of the bit diagnosis number where the cursor locates.

NO.	DATA	NO.	DATA	NO.	DATA
000	00000000	010	00000000	020	00000000
001	00000000	011	00000000	021	00000000
002	00000000	012	00000000	022	00000000
003	00000000	013	00000000	023	00000000
004	00011111	014	00000000	024	00000000
005	00000000	015	00000000	025	00000000
006	00000000	016	00000000	026	00000000
007	00000000	017	00000000	027	00000000
008	00000000	018	00000000	028	00000000
009	00011111	019	00000000	029	00000000

BIT0:
NO. 000

2) PLC SIGNAL interface

In the window of this interface, it orderly displays the state of address X0000~X0063, Y0000~Y0047, F0000~F063, G0000~G063 etc.. And it enters PLC STATE interface by pressing



key repeatedly. The signal state of PLC addresses can be viewed by pressing or



key.

In PLC STATE window, there are 2 rows at the window bottom, the 2nd row shows the meaning of a bit of an address where the cursor locates, the bit to be displayed can be positioned by pressing



or key. The 1st row shows the abbreviation of the bit address number where the cursor locates.

PLC SIGNAL					
NO.	DATA	NO.	DATA	NO.	DATA
X000	00000000	X010	00000000	X020	00000000
X001	00000000	X011	00000000	X021	00000000
X002	00000000	X012	00000000	X022	00000000
X003	00000000	X013	00000000	X023	00000000
X004	00001000	X014	00000000	X024	00000000
X005	00000000	X015	00000000	X025	00000000
X006	00000000	X016	00000000	X026	00000000
X007	00000000	X017	00000000	X027	00000000
X008	00000000	X018	00000000	X028	00000000
X009	00000000	X019	00000000	X029	00000000

BIT0:
X000

3) MACHINE SOFT PANEL

Press  key repeatedly, it enters MACHINE SOFT PANEL. In this page, it can control soft keyboard for machine. The page displays as follow:

MT SOFT PANEL		Our File: 00001	
X000.0 X axis deceleration sig	●	Y000.0 Spindle delay braking o	●
X000.1 External dwell signal	●	Y000.1 Lubricating output sign	●
X000.2 Y axis deceleration sig	●	Y000.2 Undefined	●
X000.3 Z axis deceleration sig	●	Y000.3 Spindle CW signal	●
X000.4 4th deceleration signal	●	Y000.4 Spindle CW signal	●
X000.5 Emergency stop signal	●	Y000.5 Spindle stop signal	●
X000.6 Input signal of overtra	●	Y000.6 055 output channel	●
X000.7 Switch X4 of starting	●	Y000.7 054 output channel	●
X001.0 Switch X5 of blow mode	●	Y001.0 Blow solenoid valve	●
X001.1 Tailshaft advance in pl	●	Y001.1 Material rest rise/fall	●
X001.2 Tailshaft retreat in pl	●	Y001.2 Tailshaft lock solenoid	●
X001.3 Tailshaft lock in place	●	Y001.3 Tailshaft advance/retre	●
X001.4 External cycle start si	●	Y001.4 Tailshaft advance/retre	●
X001.5 Undefined	●	Y001.5 Trimming solenoid valve	●
X001.6 Undefined	●	Y001.6 Blank pressing retreat	●
X001.7 Protective door detecti	●	Y001.7 Blank pressing advance	●

Note: It can be switched between row and row by pressing change key

4) HELP MESSAGE interface

It enters HELP MESSAGE interface by pressing  key repeatedly. The operation list, alarm list, G command list, and macro command message can be shown in this interface. As is shown in the following figure:

MDI	I S0000 T0000
HELP INFO	00001 N0000
Clear alarm: RESET (anti clash key)	
page turning: page key	
search: P key -> figure -> input key	
PRG execution: automatic/MDI mode circle start	
location/POS interface	
interface change: location key/page turning	
PRT CNT clear: cancel key+N key	
RUN TIME clear: cancel key+M key	
display time change: change key	
rel coord page	
rel coord clear: U key/W key -> cancel key	
program/PRG interface	
RUN TIME: 00:36:28	
CNC DGN	PLC SIGNAL
MT FACE	VERSION I
CNC HELP	

5) VERSION MESSAGE interface

It enters VERSION MESSAGE interface by hardware, and PLC version message can be shown in this figure:

pressing  key repeatedly. The software, in this interface. As is shown in the following



Chapter II Power ON/OFF and Safety Protection

■ Power ON

Please confirm before GH1000MC CNC system is powered on:

- The machine status is normal;
- The power supply voltage meets the requirements;
- The connection is correct and firm.

The interface is shown as below after GH1000MC is powered on:

Then, GH1000MC is self-detected and initialized. After the self-detection and initialization, the page of the present position (the relative coordinate) is displayed as below:



■ Power OFF

Please confirm before power off:

- The feeding axis of CNC is in stopping status;
- The miscellaneous function (such as the spindle, the water pump, etc) is off;
- Firstly cut off CNC power supply, and then the machine one.

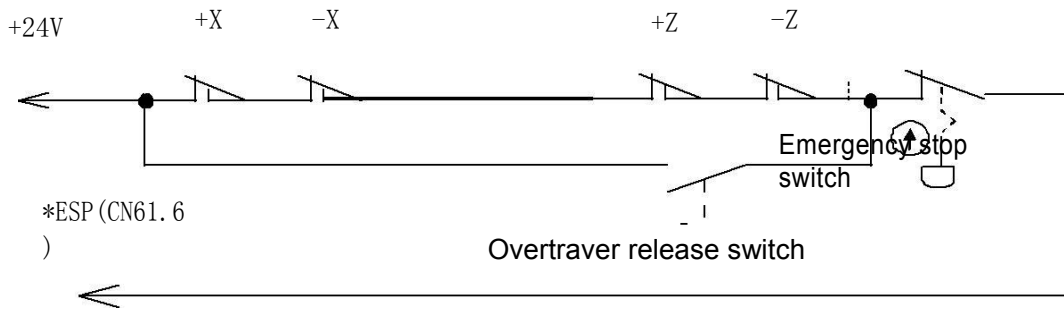
Note: About the operation of switching off the machine power supply, please refer to the user manual of the machine manufacturer.

■ Overtravel Protection

The machine must be adopted the overtravel protection measure to avoid the X ,Y and Z axes overtravel causing the machine damage.

■ Hardware Overtravel Protection

The positive and negative maximum travel positions of the X ,Y and Z axes are installed with the travel limit switch on the machine, and the connection is operated based on the following figure. And then, Bit2(EALM) and Bit3(LALM) of the status parameter No.60 must be set as 0. When the overtravel occurs, the limit switch is operated, and GH1000MC CNC system stops the movement and displays the emergency stop alarm.



When the hardware overtravel occurs, GH1000MC CNC system issues the alarm of “emergency stop”. The method of clearing the alarm of “emergency stop”: Press the overtravel release button without releasing, switch into the alarm information page, and check the alarm information. After clearing the alarm by resetting, the worktable is moved in the opposite direction (for example, move it in the negative direction if it is the positive overtravel; vise versa), and the limit switch is released.

2.3.2 Software Overtravel Protection

The software travel stroke is set by data parameter NO.10~ NO.19, they refer to machine coordinate. Bit1(LZR) of No.22 can be set if the software limit is valid before machine return zero

If the machine position (coordinate) exceeds the setting range, overtravel alarm will occur. The steps to eliminate this alarm is press RESET key to clear the alarm, then moves reversely (for positive overtravel, move out negatively; vice versa)

■ Emergency Stop Operation

During machining, due to the user programming, operation and product fault, etc, some unexpected situations may occur, and GH1000MC CNC system must be stopped immediately. In the chapter, it mainly introduces the process for GH1000MC CNC system in the emergency situation; about the machine one, please refer to the relative explanation of the machine manufacturer.

■ Resetting

When the GH1000MC CNC system is abnormally output and the coordinate axis is

moved, press  key to make the GH1000MC CNC system in the resetting status:

1. All axes movement stops;
2. M and S function output is invalid (whether automatically switch off the signals of the spindle

rotation, the lubrication and the cooling, etc after pressing  key is set by the parameter, the ladder diagram definition);

After the automatic running ends, the modal function and status remain.

■ Emergency Stop

Press the emergency stop button (when the external emergency stop signal is valid) in the dangerous or in the emergency situation during the machine running, and then CNC enters the emergency stop status, then:

The machine movement stops immediately, the output of the spindle rotation and the coolant are all switched off. Release the emergency stop button to release the alarm and CNC enters the resetting status. About the circuit connection method, please refer to the chapter 2.3.1.

Note 1: Please confirm the troubleshooting is completed before releasing the emergency stop alarm;

Note 2: Pressing the emergency stop button before power ON/OFF can reduce the electric shock upon the equipment;

Note 3: Please execute the machine zero return after the emergency stop alarm is released, so the correctness of the coordinate position can be guaranteed (if the machine isn't installed with the machine zero, the machine zero return should NOT be operated):

Note 4: Only when Bit 2 of the status parameter NO.60 is set as 0, the external emergency stop is valid.

■ **Feed hold**



During the machine tool running, press  to make the running dwell. Please pay attention to that the function can't make the running stop immediately during the thread cutting and the tapping cycle.

■ **Cut off the Power Supply**

During the machining tool running, the machine power supply should be cut off immediately in the dangerous or in the emergency situation to avoid the accident. But please pay attention to that the coordinate displayed on CNC has the big deviation with the actual position after power off, so resetting the tools should be operated.

No

The function of the buttons on GH1000MC machine tool panel is defined by PLC program (the ladder diagram), and about the function and meaning of each button, please refer to the manual of the machine tool manufacturer.

Please pay attention to that the following chapter about the relative function of the operation panel buttons is mainly for GH1000MC standard PLC program!



Press key to enter the Manual mode. In Manual mode, the manual feed, the spindle control, the override adjusting and the tool change, etc can be operated.

3.1 The Coordinate Axis

In Manual mode, two-axis manual feed and manual rapid traverse can be

3.1.1 Manual

Press  or  of    to make X axis feeding in positive or

negative direction, and the axis movement stops after releasing the button; press  or  to make Z axis feeding in positive or negative direction, and the axis movement stops after releasing the

button; press  or  to make the Y axis feeding in positive or negative direction, and the Y axis movement stops after releasing the button; press  or  to make the 4th axis feeding in positive or negative direction, and the axis movement stops after releasing the button;

During the manual feeding, press  and the indicator lamp is ON, the system enters the manual rapid traverse status.

■ **Manual Rapid Movement**

Press  of       to make the indicator lamp ON, press  or  to make X axis rapid traverse in positive or negative direction, and the axis movement stops after releasing the button; press  or  to make Z axis rapid traverse in positive or negative direction, and the axis movement stops after releasing the button; press  or  to make Y axis rapid traverse in positive or negative direction, and the axis movement stops after releasing the button; press  or  to make the 4th axis rapid traverse in positive or negative direction, and the axis movement stops after releasing the button;

During the manual rapid traverse, press  to make the indicator lamp OFF, the rapid traverse is invalid, and the the feeding is executed in manual feedrate.

Note 1: After power ON, if the reference position isn't returned and the rapid movement switch (the rapid movement button indicator lamp is ON), whether the rapid traverse feedrate is manual feedrate or rapid traverse rate is set by Bit0(ISOT) of GH1000MC the status parameter NO.012.

Note 2: In Edit/MPG mode,  key is invalid.

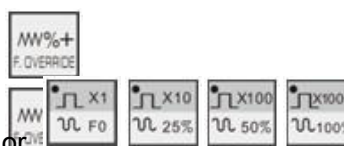
■ **Speed tune**

In Manual mode,  or  can be pressed to alter the manual feedrate override that has 16 steps. The relation of the override and the feedrate is as follows table if data parameter No.031 is set to 1260:

Feedrate override (%)	Feedrate (mm/min)
0	0
10	126
20	252
30	378
40	504
50	630
60	756

Feedrate override (%)	Feedrate (mm/min)
70	882
80	1008
90	1134
100	1260
110	1386
120	1512
130	1638
140	1764
150	1890

Note : There is about 2% error for the data in the above table.



In the manual rapid traverse, it can press the  key to modify the rapid override, and there are 4 steps of F0, 25%, 50%, 100% for the override. (F0 set by data parameter No.085)

■ Other Manual Operation

1. CCW Rotation, CW Rotation, Stop controlling



: In manual mode, press the key, the spindle CW rotate;



: In manual mode, press the key, the spindle start or stop;



2. In manual mode, press the key, the spindle decelerate rotate;

■ Spindle jog



: At the moment, the spindle is in JOG state.



Functional description: Press  to enter JOG mode, and the spindle JOG function ON/OFF is executed only when the spindle is in the state of stop.

In spindle JOG mode, by pressing



key, the spindle rotates counterclockwise for jogging;

by pressing  key, the spindle rotates clockwise for jogging. The jog time and speed are set by data parameter No.108 and No.109 respectively.

When the spindle JOG rotates,  is pressed to stop the spindle JOG rotation, the spindle brake signal is not output when the JOG rotation stops.

K10.4 is set to 1, the spindle JOG is valid in any mode. In Auto or MDI mode, the spindle is in the JOG rotation state, the program closes the spindle JOG rotation and the JOG function.

Parameter setting:

PLC parameter K104 1/0: the spindle JOG is valid in any mode/Manual, MPG, Zero return mode.

Data parameter No.208: rotary speed in spindle JOG.

3.2.3 Cooling control



: In Manual mode, press this key, the cooling is switched on/off.

Parameter setting: PLC parameter K10.1 1/0: the spindle lubricating and cooling output remains/closes in reset.

3.2.4 Lubricating control

Function description:

■ Non-automatic lubricating

DT13 =0: For non-automatic lubricating

When DIT13=0, it is lubricating turn output, by pressing the  key, the lubricating is output. And the lubricating is cancelled by pressing it again. M32 is for lubricating output, and M33 is for lubricating output cancellation.

When data parameter DIT13 > 1, it is timing lubricating output, by pressing the key,  the lubricating is output. And it is cancelled after a setting time by data parameter No.112; by executing M32, the lubricating is output, and it is cancelled after a setting time by data parameter No.112. If the setting time is not yet up, M33 is executed to cancel the lubricating output.

2. Automatic lubricating

DT13>0: For automatic lubricating, the lubricating time DT13 and lubricating interval time DT53 may be set.

After GH1000MC system is switched on, it is lubricating for a time set by DT13, then the lubricating output stops. After an interval set by DT53, the lubricating is output again, so it cycles by

sequence. In the automatic lubricating, M32, M33 codes as well as the  key are all inactive.

Parameter setting:

PLC parameter: K10.1 1/0: the spindle lubricating/cooling output remains/closes in reset.

PLC parameter:K16.2 1/0: whether the lubricating outputs in power-on when the automatic lubricating is valid.

PLC data: DT53 automatic lubricating interval time (ms)

PLC data: DT13: automatic lubricating output time (ms)

DT15: M execution duration(ms)

DT13: lubricating start time (0-60000ms)(0:lubricating time is not limited)

3.2.5 Manual tool change



: In Manual mode, by pressing this key, the spindle tools is loosed/clamping

3.2.6 Spindle override

In Manual mode, if the spindle speed is controlled by analog voltage output, the spindle speed may be overridden.



By pressing in Spindle Override keys, the spindle speed can be changed by real-time adjusting of the spindle override that has 8 steps of 50% ~ 120% .

Chapter IV MPG/Single Step Operation

In MPG/single step mode, the machine is moved based on the selected incremental value.

NOTE!

The function of the buttons on GH1000MC machine tool panel is defined by PLCprogram (the ladder diagram), and about the function and meaning of each button, please refer to the manual of the machine tool manufacturer.

Please pay attention to that the following chapter about the relative function of the operation panel buttons is mainly for GH1000MC standard PLC program!

■ Single Step Feeding

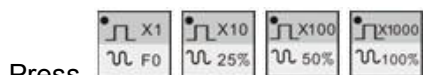
Set the system parameter No.001 Bit3 to 0, and press



key to enter the STEP working mode, it displays as follows:

STEP		S0000 T0000	
MONITOR		00003 N0000	File location
(MACHINE)		(ABS)	G01 G94 G98 G21
X	2.000	X	2.000
Y	6.000	Y	6.000
Z	1.000	Z	1.000
			M33 M41 M13 M30
			S0200
			F000000 mm/min
			300 mm/min
PRG		Comprehensive	
00003 :		STEP W : 0.001	
G00 X2 Y2 Z2 :		FEEDRATE 100%	
G01 X3 Y4 Z5 :		RAPID: 100%	
G01 X5 Y1 Z6 :		S OVRD: 100%	
G01 X10 Y9 Z7 :		PART CNT 0000/0002	
G00 X5 Y8 Z22 :		CUT TIME 000:00:19	
		RUN TIME: 09:15:55	
Ln:1			
REL		ABS	ALL
		MONI	

4.1.1 Increment selection



Press key to select the movement increment, and the movement increment is displayed on the page. When BIT7 (SINC) of PLC status parameter K016 is set as 1,

the step length value  is invalid; when BIT7 is 0,     all are valid.

If  key is pressed, the page is shown as below:



■ Selecting the Movement Direction



Press once, X axis is fed once based on the single step increment in negative

or positive direction; press  or , Z axis is fed once based on the single step increment

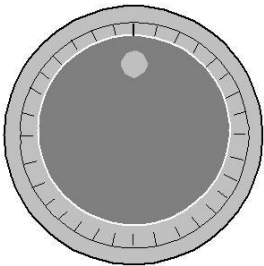
in negative or positive direction; press  or , Y axis is fed once based on the single step increment in negative or positive direction.

■ **MPG (manual pulse generator) feeding**

Set the BIT3 of the system parameter No.001 to 1, and press  key to enter the MPG mode, it displays as follows:



MPG outline is shown as the following figure:



MPG outline figure

4.2.1 Increment selection

Press     key to select the movement increment and it will be displayed

on the page. When BIT7 (SINC) of PLC K016 parameter is set as 1,  step width value is

invalid; when BIT7 is 0,     all are valid. If  key is pressed, the page is shown as below:



■ Selecting the Movement Axis and the Direction

In MPG mode, press  ,  ,  ,  key to select the corresponding axis.

MPG feeding direction is set by MPG rotation direction. Normally, MPG CW rotation is the positive feeding and CCW rotation is the negative feeding. If sometimes MPG CW rotation is negative feeding and CCW rotation is positive feeding, the signals A and B at MPG end can be exchanged or the feeding direction of MPG rotation is selected by BIT0 of parameter No.013.

■ Other Operation

1) Spindle CCW/CW rotation, stop



: In MPG/single step mode, press the key to make the spindle CW rotation;



■ In MPG/single step mode, press the key to make the spindle stop;

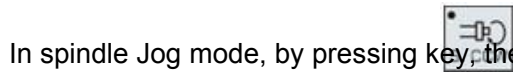


■ In MPG/single step mode, press the key to make the spindle CCW rotation;

2) Spindle Jog



: at the moment, the spindle is in JOG working mode.



In spindle Jog mode, by pressing key, the spindle rotates counterclockwise for jogging; by pressing



key, the spindle rotates clockwise for jog. The jogging time and speed are set by data parameter No.208 respectively. The concrete is referred to Chapter 3.2.2.

3) Cooling control

Refer to OPERATION, Chapter 3.2.3

4) Lubricating control

Refer to OPERATION, Chapter 3.2.4

5) Manual tool change



: In MPG/Step mode, press it to execute the tool change orderly.

6) Spindle override tune

In MPG/Step mode, if the spindle speed is controlled by analog voltage output, the spindle speed may be overridden.



By pressing in Spindle Override keys, the spindle speed can be changed by real-time adjusting of the spindle override that has 8 steps of 50% ~ 120%.

■ Explanation items

The relation between MPG scale and the machine movement amount is shown as the following list:

	Moving amount of each MPG scale			
MPG increment	0.001	0.01	0.1	1
Specified coordinate value	0.001mm	0.01mm	0.1mm	1mm

(Taking example of the least input increment 0.001mm)

Note 1: MPG increment is relative with the current metric/inch system and the system least input increment.

Note 2: The speed of MPG rotation should NOT be higher than 5r/s; otherwise, the scale value may NOT be complied with the movement amount.

Chapter V MDI Operation

In MDI mode, setting the parameters, inputting the single block and executing the single block can be performed.

NOTE!

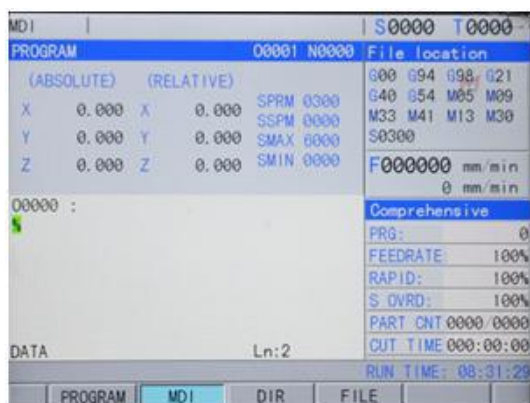
The function of the buttons on GH1000MC machine tool panel is defined by PLCprogram (the ladder diagram), and about the function and meaning of each button, please refer to the manual of the machine tool manufacturer.

Please pay attention to that the following chapter about the relative function of the operation panel buttons is mainly for GH1000MC standard PLC program!

■ MDI the Block

Select MDI mode, enter the program-> MDI program page, input one block G50 X50 Z100, the operation steps are as below:

- Press  key to enter MDI mode;
- Press  key ( or  or press  many times if needed) to enter PRG STATE window:



- 3) Input the address key  and the digit keys  and  in order;
- 4) Input the address key  and the digit keys ,  and  in order;
- 5) Input the address key  and the digit keys  and  in order.
- The window is shown as follows after above operations are completed(can input 4 block and display 6 block):



■ Executing the Block

After inputting the block, press  key to display the following page:



After inputting the block, press  to confirm and then press  to execute the inputted

block. During running, press ,  and emergency stop buttons to stop the program running.

Note: The subprogram calling code(M98 P ; etc) are invalid in MDI mode.

■ Setting the Parameters

In MDI mode, enter the parameter interface to rewrite the parameter values; about the details, refer to chapter 10 in the section.

■ Rewriting the Data

In the PRG STATE window of MDI mode, if there is an error during words inputting,  is pressed to cancel the display or  is pressed to clear all the input, then re-input the correct ones.

5.5 Other Operation

1) Adjusting the spindle override

In MPG/STEP mode, the spindle speed can be adjusted when the spindle speed is controlled by outputting the analog voltage.

The spindle speed can be changed by adjusting the spindle override by pressing  which can realize the spindle overrides 50% ~ 120%, total 8-level real-time adjustment.

2) Adjusting the rapid override

The rapid traverse feedrate can be adjusted by pressing     keys and the rapid traverse rate of 4-level real-time adjusting can be realized.

3) Feedrate override is available.

 
In MDI mode, by pressing , the actual speed real-time adjusting of 0 ~ 150% feedrate by F code can be done by the override that has 16 levels

Chapter VI Editing and Managing the Programs

In Edit mode, the program can be set, opened, rewritten, copied and deleted, and dual-channel communication between CNC and PC can be realized.

To avoid the program is rewritten and deleted by accident, the program switch is set in GH1000MC CNC system. Before editing the program, the program switch must be opened; about setting the program switch, refer to chapter 10.1.1 of the section.

For convenient management, the multi-level user authority management is set in GH1000MC CNC system. The program switch ON/OFF and editing the program can be executed only when the user is with the operation authority above level 4 (level 4 or 3, etc). About the operation for each level authority, refer to chapter 10.1.2.

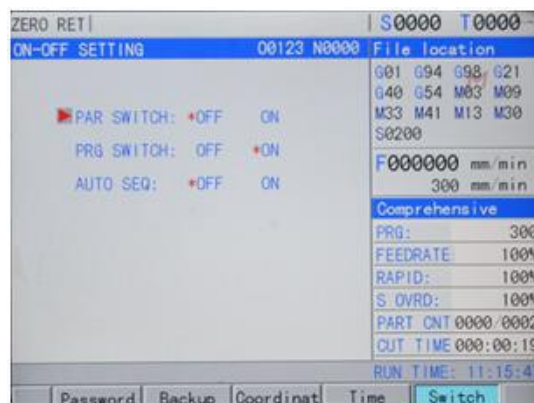
■ Setting the Program

■ Generating the Block No.

In the program, the block number can be edited and it's OK if it isn't edited. And the program is executed based on the input order. (Except that it is calling.).

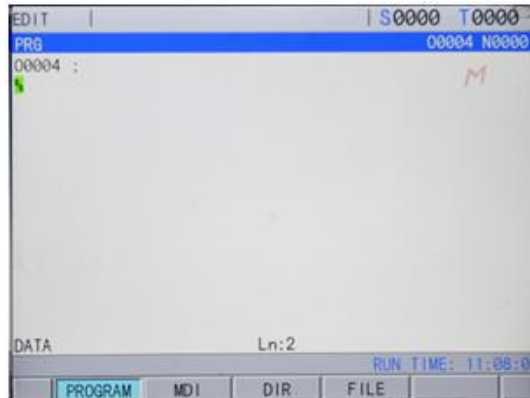
On the setting switch page, when "automatic sequence number" is OFF, CNC doesn't generate the block number automatically, while the block number can be input manually during programming.

On the setting switch page, when "automatic sequence number" is ON, CNC generates the block number automatically. During editing, press  key to automatically generate the block number of the next block, and the incremental value of the block number is set by the data parameter No. 042 of CNC data (about setting the switch of the automatic sequence number, refer to chapter 10.1.1 of the section).



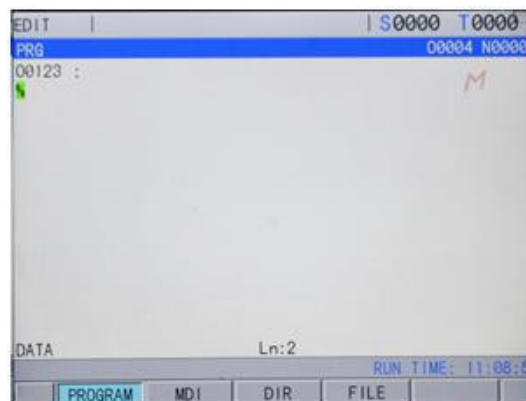
■ Inputting the Program Content

1. Press  key to enter the Edit mode and press  key to enter the program page set.



- On the program content page, press the address key , and then input the numerical keys ,  and  on the pop-up dialogue box (setting O0001 program is taken as one example, and the leading zero before inputting the program name can be omitted), the display is as below:

- Press  key to set a new program, the display is as below:



4. Input the edited part program one by one, the character will be displayed on the screen immediately as it is input (as for compound key, press this key repeatedly for alternate input), after a

block is finished, press  key to terminate it.

5. Other blocks input may be finished by step 4 above.

■ Movement of cursor

1. Press  key to enter the Edit mode, then press  key to enter the PRG CONTENT window;

2) Press  key, the cursor shifts a row upward; if the number of the column where the cursor locates is over the total columns of the previous row, the cursor moves to the previous block

end (at “,” sign) after  key is pressed;

3) Press  key, the cursor shifts a row downward; if the number of the column where the cursor locates is over the total columns of the next row, the cursor moves to the next block end (at “,”

sign) after the  key is pressed;

4) Press  key, the cursor shifts a column to the right; if the cursor locates at the row end, it moves to the head of the next block;

5) Press  key, the cursor shifts a column to the left; if the cursor locates at the row head, it moves to the end of the next block;

■ Press  key to window upward

■ Press  key to window downward,

■ Searching No. of character and line

Searching a character: To search for the specified character upward or downward from the cursor current location

The steps of finding is as follows:

1) Press  key to enter Edit mode;

2) Press  key to enter the PRG CONTENT window;

■ Input the characters to be searched , the characters over the 10th byte will be ignored.

1. Press  key ( or  is determined by the location of the character searched to the character where the cursor locates), it displays as follows:



5) After the finding, the CNC system is still in FIND state, press  or  key again, the next character can be found.

Note 1: If the character is not found, the searching character will disappear

Note 2: During the searching, it doesn't search the characters in the called subprogram, and the character in subprogram is searched in subprogram.

Note 3: The system cannot search and scan the character in macro edit mode.

Searching a line: Put the cursor rapidly move to specified line of program

The steps of finding is as follows:

2. Press  key to enter Edit mode;

3. Press  key to enter the PRG CONTENT window;

■ Input the line No. to be searched

4) Press  key, the cursor will skip to specified line

6.1.5 Inserting a character

Steps:

- Select the PRG CONTENT window in Edit mode;
- Press  key to enter the INS mode (the cursor is an underline), the window is as follows:
- Input the character to be inserted

■ Deleting a character

Steps:

- 1) Enter the PRG CONTENT window in Edit mode;
- 2) Move the cursor to the location where you want to delete, press  key to delete the character where the cursor locates;

6.1.7 Altering a character

Steps:

- 1) Enter the PRG CONTENT window in Edit mode;
- 2) Move the cursor to the location where you want to alter, press  key to alter the character instead of the input content

6.1.8 Deleting a single block

Steps:

- Select the PRG CONTENT window in Edit mode;
- Move the cursor to the head of the block to be deleted (column 1), press  key,
- And then press  key to delete a single block

6.1.9 Deleting blocks

Steps:

- Select the PRG CONTENT window in Edit mode;
- Move the cursor to head line of a block that you want to delete
- Input the sequence No. of the last block that you want to delete
- 4) Press  key, the blocks among cursor and marked address will be deleted

6.1.10 Deleting a segment

Steps:

1. Select the PRG CONTENT window in Edit mode;
2. Move the cursor to the 1st character of a block that you want to delete

- Input the last character of the block that you want to delete

- 4) Press  key, the segment among cursor and marked address will be deleted

6.1.11 Copying a single block

Steps:

- 1) Select the PRG CONTENT window in Edit mode;
- 2) Move the cursor to head line of a block that you want to copy

- 3) Press  key first, then press , copy the block where the cursor located in

6.1.12 Copying blocks

Steps:

- 1) Select the PRG CONTENT window in Edit mode;
- Move the cursor to 1st character of a block that you want to copy
- Input the sequence No. of the last block that you want to copy

- 4) Press  key, the blocks among cursor and inputted character will be copied

6.1.13 Copying a segment

Steps:

1. Select the PRG CONTENT window in Edit mode;
2. Move the cursor to the 1st character of a block that you want to delete

- Input the last character of the block that you want to delete

- 4) Press  key, the segment among cursor and inputted character will be copied

6.1.14 Pasting a single block

Steps:

- 1) Select the PRG CONTENT window in Edit mode;
- 2) Move the cursor to location of program that you want to paste

- 3) Press  key, insert the last copy program content before cursor to finish paste operation

■ Deleting program

6.2.1 Deleting a program

Steps:

- Select the PRG CONTENT window in Edit mode;
- Press address key  , number key  ,  ,  ,  by sequence (by program O0001);
- Press  key, program O0001 will be deleted.

6.2.2 Deleting all programs

Steps:

- 1) Select the PRG CONTENT window in Edit mode;
- 2) Press address key  , symbol key  , number key  ,  ,  ,  ;
1. Press  key, all the programs will be deleted.

■ Selecting a program

When there are multiple programs in CNC system, a program can be selected by the following 3 methods:

6.3.1 Search

- Select Edit or Auto mode;
- Press  key to enter the PRG CONTENT window;
- 3) Press address key  and key in the program No.;
- 4) Press  or  key, or press  key in Auto mode, the searched program will be displayed. If the program does not exist, an alarm will be issued by CNC.

Note: In step 4, if the program does not exist in Edit mode, a new program will be created by CNC system

after  key is pressed.

■ Scanning

- Select Edit or Auto mode;

- Press  key to enter the Program window;

- 3) Press address key  ;

- 4) Press  or  key to display the next or previous program;

- 5) Repeat step 3 and 4 to display the saved programs one by one.

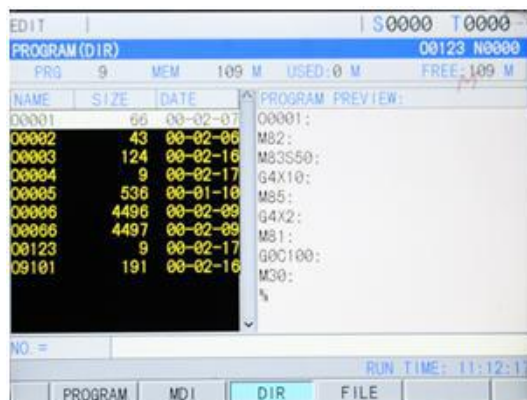
■ Cursor

- Select Auto mode (**must be in non-running state**);

- Press  key to enter the PRG LIST window;



- 3) Press ,  , key to move the cursor to the program name to be selected
(Program content will change as the cursor moves);



- 4) Press  key or  .

■ Renaming a program

1. Select the PRG CONTENT window in Edit mode;
2. Press address key  , and key in the new program name;
3. Press  key.

■ Copy a program

To save the current program to a location:

1. Select the PRG CONTENT window in Edit mode;
2. Press address key  , and key in the new program No.;
3. Press  key.

■ Program management

6.6.1 Program list

In non-Edit mode, press  key to enter the PRG LIST window. In this window, it lists the program names saved in CNC system, and it can display maximum 10 names in a window, if the programs saved exceed 10, it may press   key to display the other program list.



NAME	SIZE	DATE	PROGRAM PREVIEW
00001	66	00-02-07	O0001;
00002	43	00-02-06	M82;
00003	124	00-02-16	M83S50;
00004	9	00-02-17	G4X10;
00005	536	00-01-10	M85;
00006	4496	00-02-09	G4X2;
00006	4497	00-02-09	M81;
00123	9	00-02-17	G0C100;
09101	191	00-02-16	M30;

1) Open program

Open specified program :  + No. + 

In Edit mode, it will creat program if program No. is not exit.

2) Delete program

1. In Edit mode, press  to delete program specified by

cursor 2.In Edit mode,press  + No. +  or No. + 

6.6.2 Part-Prg number

It shows the total number of the part programs (up to 400) that can be saved in CNC system and the current part programs number that have been saved at present.

■ Memory size and used capacity

They show the total memory capacity (56M) of the CNC and the current capacity that has been occupied.

CHAPTER 7 AUTO OPERATION

Note !

The key functions of GH1000MC machine panel are defined by PLC program (ladders), please refer to the materials by the machine builder for their significance.

Please note that the following description for the keys function in this chapter is based on GH1000MC standard PLC program!

7.1 Automatic run

7.1.1 Selection of the program to be run

1. Searching method

- Select the Edit or Auto mode;
- Press  key to enter the PRG CONTENT window;
- Press the address key , and key in the program No.;
- Press  or  key, the program retrieved will be shown on the screen, if the program doesn't exist, an alarm will be issued.

2. Scanning method

1. Select the Edit or Auto mode;
2. Press  key to enter the PRG CONTENT window;
3. Press the address key ;
4. Press the  or  key to display the next or previous program;
5. Repeat the step 3, 4 above to display the saved program one by one.

3. Cursor method

- Select the Auto mode (in non-run state);
- b) Press  key to enter the PRG LIST window (press  or  key if needed);
- Press , ,  or  key to move the cursor to the name of the program to be selected;
- Press  key.

■ Start of the automatic run

- Press  key to select the Auto mode;

- Press  key to start the program, and the program automatically runs.

Note: Since the program execution begins from the block where the cursor locates, before pressing the



key, make a check whether the cursor is located at the block to be executed.

7.1.3 Stop of the automatic run

* Stop by code (M00)

1. M00

After the block containing M00 is executed, the auto run is stopped. So the modal function and state are all reserved. Press the  key or the external run key, the program execution continues.

2. M01

Press  and the optional stop indicator is ON and the function is valid. After the block with M01 is executed, the system stops the automatic run, the modal function and the state are saved.

Press  or the external run key, and the program continuously runs.

* Stop by a relevant key

1. In Auto run, by pressing  key or external dwell key, the machine keeps the following state:

- The machine feed slows down to stop;
- The modal function and state are reserved;
- The program execution continues after pressing the  key.

2. Stop by Reset key

- All axes movement is stopped.
- M, S function output is inactive (the automatic cut-off of signals such as spindle CCW/CW,

lubricating, cooling by pressing  key can be set by the parameters)
(3) Modal function and state is held on after the auto run.

3. Stop by Emergency stop button

If the external emergency button (external emergency signal active) is pressed under the dangerous or emergent situation during the machine running, the CNC system enters into emergency state, and the machine moving is stopped immediately, all the output (such as spindle rotation, cooling) are all cut off. If the Emergency button is released, the alarm is cancelled and CNC system enters into reset mode.

4. Switching operation mode

When Auto mode is switched to the Machine zero, MPG/Step, Manual, Program zero mode, the current block “dwells” immediately; when the Auto mode is switched to the Edit, MDI mode in Auto

mode, the “dwell” is not displayed till the current block is executed.

Note 1: Ensure that the fault has been resolved before cancelling the emergency alarm.

Note 2: The electric shock to the device may be decreased by pressing the Emergency button before power on and off.

Note 3: The Machine zero return operation should be performed again after the emergency alarm is cancelled to ensure the correctness of the position coordinates (but this operation is forbidden if there is no machine zero in the machine).

Note 4: Only the BIT2 (EALM) of the bit parameter No.215 is set to 0, could the external emergency stop be active.

7.1.4 Automatic run from an arbitrary block

Press  key to enter the Edit mode, press  key to enter the Program interface, then press  or  key to enter the PRG CONTENT window:

1. Move the cursor to the block to be executed (for example, move the cursor to the 3rd row head if it executes from the 3rd row);



■ If the mode (G, M, T, F code) of the current block where the cursor locates is defaulted and inconsistent with the running mode of this block, the corresponding modal function should be executed to continue next step.

■ Press  key to enter the Auto mode, then press  key to start the execution.

7.1.5 Adjustment of the feedrate, rapid rate

In Auto mode, the running speed can be changed by adjusting the feedrate override, rapid override. It doesn't need to change the settings of the program and parameter.

* Adjustment of the feedrate override



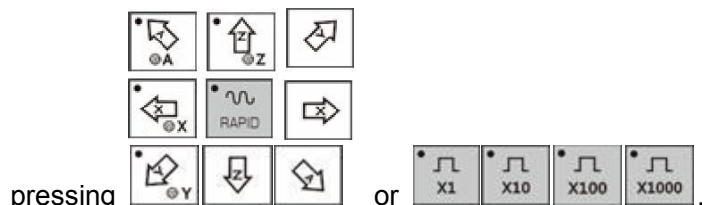
Press , 16-level real time feedrate can be obtained.

Note 1: The actual feedrate value is specified by F in feedrate override adjustment;

Note 2: Actual feedrate= value specified by F × feedrate override

* Adjustment of rapid override

It can realize the F0, 25 %, 50 %, 100 % 4-level real time rapid override adjustment by



Note 1: The rapid traverse speeds of all axis are set by the CNC parameter No.090~No.094 respectively;

Actual rapid traverse rate = value set by parameter × rapid override

Note 2: When the rapid override is F0, the min. rapid traverse rate is set by bit parameter No.085.

Note 3: For spinning machine, only after finish returning zero can run program automatically .

7.3 State of running

7.3.1 Single block run

In Auto mode, the methods for turning on single block switch are as follows:

Method 1: Press the  key to make the single block indicator  in panel state area to light up, it means that the single block function has been selected;

In Single mode, when the current block execution is finished, the CNC running stops; if next

block is to be executed, it needs to press the  key again, then repeat this operation till the whole program is finished.

Note 1: The single block stops at the mid point of G28 code.

Note 2: While the subprogram calling (M98__), or subprogram calling return (M99) is being executed, the single block is inactive. But it is active except for N, O, P addresses in the block that contains M98 or M99 code.

7.3.2 Dry run

Before the program is to be executed automatically, in order to avoid the programming errors, it may select the Dry run mode to check the program. In Auto mode, the methods for turning on the Dry run switch are as follows:

Press the  key to make the Dry run indicator in panel state area to light up, it means that the dry run mode has been selected;

In Dry run mode, the machine feed and miscellaneous functions are both active (as machine lock, MST lock are both OFF), that means the dry run switch has nothing to do with the machine feeding, MST functions, so the feedrate by program is inactive and the CNC system runs by data parameter No.082

7.3.3 Machine lock

In Auto mode, the turning on method of machine lock switch is as follows:

Method 1: Press the  key to make the Machine Lock indicator  in panel state area to light up, it means that it has entered the machine lock state;

The machine lock and MST lock are usually used together to check the program. While as in the machine lock mode:

- The machine carriage doesn't move, the "MACHINE" in the INTEGRATED POS window of the Position interface doesn't vary too. The RELATIVE POS and ABSOLUTE POS, DIST TO GO are refreshed continuously, which is the same as that the machine lock switch is OFF.
- M, S, T commands can be executed normally.

7.3.4 MST lock

In Auto mode, the turning on of MST lock switch is as follows:

Method 1: Press the  key to make the MST Lock indicator  in panel state area to light up, it means that it has entered the MST lock state;

The machine carriage moves without the M, S, T code being executed. The machine lock and MST lock are usually used together to check the program.

Note: When the MST lock is active, it takes no effect to the execution of M00, M30, M98, M99.

7.3.5 Block skip

If a block in program is not needed to be executed and not to be deleted, this block skip function can be used. When the block is headed with "/" sign and Block skip indicator lights up (panel key active or external skip input active), this block is skipped without execution in Auto mode.

In Auto mode, the turning on of Block skip switch is as follows:

Method 1: Press the  key to make the Block skip indicator  in panel state area to light up;

Note: While the block skip switch is off, the blocks headed with "/" signs are executed normally in Auto mode.

■ Other operations

1. In Auto mode, press  key to switch on/off the cooling;

2. Press any of the        keys to switch the operation modes;

- Press the  key to reset this CNC system.
- Automatic lubricating operation (Refer to **Volume II Operation, Chapter 3**).

CHAPTER 8 ZERO RETURN OPERATION

Note!

The key functions of this GH1000MC machine panel are defined by PLC program (ladders), please refer to the manuals by the machine builder for their significance.

Please note that the following description for the panel key functions in this chapter is based on the GH1000MC standard PLC program!

8.1 Machine Zero return

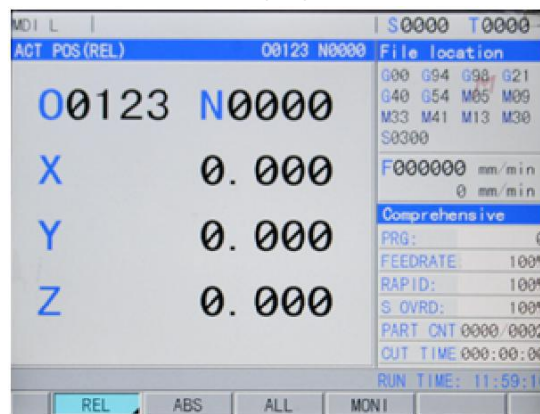
■ Machine Zero (machine reference point)

The **machine coordinate system** is a reference coordinate system for CNC coordinate operation. It is an inherent coordinate system of the machine. The origin of the machine coordinate system is called machine zero (or mechanical reference point). It is defined by the zero or zero return switch fixed on the machine. Usually this switch is fixed at the positive stroke point of X or Z axis.

8.1.2 Machine Zero return steps



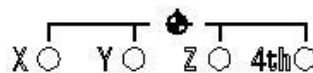
1. Press  key, it enters the Machine zero mode, the bottom line of the window displays “MACHINE ZERO”, as the following figure shows:



2. Press , ,  or  key to return to the machine zero of X, Z or Y axis;

3. The machine axis returns to the machine zero via the deceleration signal, zero signal detection.

At the machine zero, the axis stops, and the corresponding machine zero return completion indicator lights up.



Machine zero return completion indicator

Note 1: If there is no machine zero on the machine, machine zero operation is forbidden;

Note 2: The machine zero finish indicator is gone out on condition that:

- 1) The axis is moved out from machine zero;
- 2) CNC is powered off.

Note 3: After the machine zero operation, the tool length compensation is cancelled by CNC;

Note 4: Parameters related to machine zero return are referred to Volume III INSTALLATION and CONNECTION.

Note 5: After the machine zero return is executed, the original workpiece coordinate system is set again

8.2 Other operations in zero return

1. Press  key, the spindle rotates counterclockwise;
2. Press  key, the spindle stops;
3. Press  key, the spindle rotates clockwise;
4. Press  key, the cooling is switched ON or OFF;
 - Lubricating control(refer to OPERATION, Chapter 3);
- Press  key, the tool change is executed;
- Tune the spindle override;
- Tune the rapid override;
- Tune the feedrate override.

CHAPTER 9 DATA SETTING, BACKUP and RESTORE

9.1 Data setting

9.1.1 Switch setting

In SWITCH SETTING window, the ON-OFF state of PARM SWT (parameter switch), PROG SWT (program switch), AUTO SEG (auto sequence No.) can be displayed and set, as is shown in following figure:



■ Press  key to enter the Setting interface, then press  or  key to enter the SWITCH SETTING window;

■ Press  or  key to move the cursor to the item to be set;

3. Press  and  key to shift the ON-OFF state: press  key, “*” moves to the left to

set the switch for OFF, press  key, “*” moves to the right to set the switch for ON. Only the PARM SWT is set for ON, could the parameter be altered; so are PROG SWT and AUTO SEG.

Note: When the PARM SWT is shifted from “OFF” to “ON”, an alarm will be issued by CNC system. By

pressing the  ,  key together, the alarm can be cancelled. If the PARM SWT is shifted again, no alarm is issued. For security it should set the PARM SWT for “OFF” after the parameter alteration is finished.

9.1.2 Graphic display

Press  key to enter the path window



Graphic parameter meaning

Coordinate system setting: 6 types of graphic paths can be displayed in this GH1000MC CNC system depending on the front or rear tool post coordinate system

A: Graphic scaling up and down

In Graphic window, the graphic path can be scaled up and down by the keys  ,  in the edit keypad.

B: The START, STOP and CLEAR of the graphic path

In Graphic window, press the  key once, it starts the drawing; press the  key once, it stops drawing; press  key once, it clears the current graphic path.

C: Move of graphic path

In Graphic window, it may press the direction keys to move the graphic path

9.1.3 Parameter setting

By the parameter setting, the characteristics of the driver and machine can be adjusted. See Appendix 1 for their significance.

Press  key to enter the Parameter interface, then press  or  key to window the parameter interface, as is shown in the following figure:

ZERO RET		S0000 T0000	
STATUS PARAMETER		00123 N0000	
NO.	DATA	NO.	DATA
001	00001000	011	00000000
002	00000000	012	01110001
003	00010100	013	11000001
004	00000000	014	00000000
005	00011000	015	00000000
006	00001001	016	10110010
007	00000000	017	00000101
008	00001101	018	00100000
009	10011111	019	11010100
010	00001000	020	00000100
021	00000111	022	10000001
023	00000000	024	00000000
025	00000000	026	00000000
027	00000000	028	00000000
029	00110000	030	00000000
**** SPTY SCHW **** INI			
BIT0 (0: In mm 1: In inches) input			
NO.001			
RUN TIME: 11:17:49			
BITPAR NUNPAR PITCH COM USER PARA			

1. Alteration of the bit parameter

■ Byte alteration

Turn on the parameter switch;

Enter the MDI mode;

Move the cursor to the parameter No. to be set:

Method 1: Press  or  key to enter the window containing the parameter to be set, press  or  key to move the cursor to the No. of the parameter to be set;

Method 2: Press address key , key in parameter No., then press  key.

Key in the new parameter value;

Press  key, the parameter value is entered and displayed.

For security, the PARM SWT needs to be set for OFF after all parameter settings are finished.

Example:

Set the bit parameter No.004 Bit (DECI) to 1, and the other bits unchanged.

Move the cursor to No.004, input 01100000 by sequence in the prompt row, the display is as follows:

MDI RET		S0000 T0000	
STATUS PARAMETER		00123 N0000	
NO.	DATA	NO.	DATA
001	00001000	011	00000000
002	00000000	012	01110001
003	00010100	013	11000001
004	00000000	014	00000000
005	00011000	015	00000000
006	00001001	016	10110010
007	00000000	017	00000101
008	00001101	018	00100000
009	10011111	019	11010100
010	00001000	020	00000100
021	00000111	022	10000001
023	00000000	024	00000000
025	00000000	026	00000000
027	00000000	028	00000000
029	00110000	030	00000000
SCHW ISC **** SPTY SCHW **** INI			
BIT0:Retention			
NO.004 = 01100000			
RUN TIME: 11:19:38			
BITPAR NUNPAR PITCH COM USER PARA			

Press  key to finish the parameter alteration. The window is shown as follows:

MDI RET		S0000 T0000	
STATUS PARAMETER		00123 N0000	
NO	DATA	NO	DATA
001	00001000	011	00000000
002	00000000	012	01110001
003	00010100	013	11000001
004	01100000	014	00000000
005	00011000	015	00000000
006	00001001	016	10110010
007	00000000	017	00000101
008	00001101	018	00100000
009	10011111	019	11010100
010	00001000	020	00000100
SCW ISC **** **			
BIT0:Retention			
NO.004 =			
RUN TIME: 11:19:56			
BITPAR	NUNPAR	PITCH COM	USER PARA

■ Alteration by bit:

1. Turn on the parameter switch;
2. Enter the MDI mode;
3. Move the cursor to the No. of the parameter to be set;

Method 1: Press  or  key to enter the window of the parameter to be set, press  or  key to move the cursor to the No. of the parameter to be set;

Method 2: Press address key , key in parameter No., then press  key.

- 4) Press  key to skip to a bit of the parameter, and the bit is backlighted. Press  or  key to move the cursor to the bit to be altered, then key in 0 or 1;
- 5) After all parameters setting is finished, the PARM SWT needs to be set for OFF for security.

Note: After entering a bit of the parameter, press  key, it may skip out of the bit and back to the parameter No..

Example:

Set the BIT5 of the bit parameter No.004 to 1, and the other bits unchanged.

Move the cursor to “No.004” by the steps above, press  key to skip to a bit of the parameter as follows:

MDI RET		S0000 T0000	
STATUS PARAMETER		00123 N0000	
NO	DATA	NO	DATA
001	00001000	011	00000000
002	00000000	012	01110001
003	00010100	013	11000001
004	01100000	014	00000000
005	00011000	015	00000000
006	00001001	016	10110010
007	00000000	017	00000101
008	00001101	018	00100000
009	10011111	019	11010100
010	00001000	020	00000100
021	00000111	022	10000001
023	00000000	024	00000000
025	00000000	026	00000000
027	00000000	028	00000000
029	00110000	030	00000000
SCW ISC **** * * * * * * * * * *			
BIT7:(0: mm 1: inc)output			
NO.004 =			
RUN TIME: 11:20:53			
BITPAR NUNPAR PITCH COM USER PARA			

Move the cursor to “BIT5” by pressing  or  key as follows:

MDI RET		S0000 T0000	
STATUS PARAMETER		00123 N0000	
NO	DATA	NO	DATA
001	00001000	011	00000000
002	00000000	012	01110001
003	00010100	013	11000001
004	01100000	014	00000000
005	00011000	015	00000000
006	00001001	016	10110010
007	00000000	017	00000101
008	00001101	018	00100000
009	10011111	019	11010100
010	00001000	020	00000100
021	00000111	022	10000001
023	00000000	024	00000000
025	00000000	026	00000000
027	00000000	028	00000000
029	00110000	030	00000000
SCW ISC **** * * * * * * * * * *			
BIT5:Retention			
NO.004 =			
RUN TIME: 11:22:04			
BITPAR NUNPAR PITCH COM USER PARA			

Input “0” to finish the alteration.

MDI RET		S0000 T0000	
STATUS PARAMETER		00123 N0000	
NO	DATA	NO	DATA
001	00001000	011	00000000
002	00000000	012	01110001
003	00010100	013	11000001
004	01100000	014	00000000
005	00011000	015	00000000
006	00001001	016	10110010
007	00000000	017	00000101
008	00001101	018	00100000
009	10011111	019	11010100
010	00001000	020	00000100
021	00000111	022	10000001
023	00000000	024	00000000
025	00000000	026	00000000
027	00000000	028	00000000
029	00110000	030	00000000
SCW ISC **** * * * * * * * * * *			
BIT5:Retention			
NO.004 =			
RUN TIME: 11:22:18			
BITPAR NUNPAR PITCH COM USER PARA			

■ Altering data parameter and screw-pitch parameter

1. Turn on the parameter switch;
2. Enter the MDI mode;
3. Move the cursor to the No. of the parameter to be set;
4. Key in the new parameter value;
- 5) Press  key, the value is entered and displayed;
- 6) After all parameters setting is finished, the PARM SWT needs to be set for OFF for security.

Explanation: The screw-pitch parameter can only be altered under the 2 level password authority.

Example 1: set the data parameter No.000 to 2.

Move the cursor to “No.000” by the steps above, key in “2” by sequence in the cue line as follows:

MDI RET		S0000 T0000	
NUM PARAMETER		00123 N0000	
NO	DATA	NO	DATA
000	1 010	-9999.0000	020 0.0000
001	1 011	9999.0000	021 0.0000
002	1 012	-9999.0000	022 0.0000
003	1 013	9999.0000	023 0.0000
004	1 014	-9999.0000	024 0.0000
005	1 015	9999.0000	025 0.0000
006	1 016	-9999.0000	026 0.0000
007	1 017	9999.0000	027 0.0000
008	1 018	-9999.0000	028 0.0000
009	1 019	9999.0000	029 0.0000
MIN 1 MAX 65536			
X multiplication coefficient (CMR)			
NO.000 = 2			
RUN TIME: 11:22:59			
BITPAR		NUMPAR PITCH COM USER PARA	

Press  key to finish the alteration. The window is shown as follows:

WHEEL		S0000 T0000	
NUM PARAMETER		00123 N0000	
NO	DATA	NO	DATA
000	2	010	-9999.0000
001	1	011	9999.0000
002	1	012	-9999.0000
003	1	013	9999.0000
004	1	014	-9999.0000
005	1	015	9999.0000
006	1	016	-9999.0000
007	1	017	9999.0000
008	1	018	-9999.0000
009	1	019	9999.0000
MIN 1 MAX 65536			
X multiplication coefficient (CMR)			
NO.000			
RUN TIME: 11:24:06			
BITPAR		NUMPAR	
		PITCH COM USER PARA	

Example 2: set X value of the screw-pitch parameter No.000 to 12, Z axis value of that to 30.

Move the cursor to screw-pitch parameter No.000 by the steps above, key in “X12” by sequence

in the cue line. Press  key to finish the alteration.

MDI L		S0000 T0000	
PITCH COMP		00123 N0000	
NO	X	Y	Z
000	12	0	0
001	0	0	0
002	0	0	0
003	0	0	0
004	0	0	0
005	0	0	0
006	0	0	0
007	0	0	0
008	0	0	0
009	0	0	0
010	0	0	0
011	0	0	0
NO.000 =			
RUN TIME: 11:25:37			
BITPAR		NUMPAR	
		PITCH COM USER PARA	

The same as above, key in “Z30” by sequence in the cue line, press

DATA
INPUT

key to finish the alteration. The window is as follows:

MDI L		S0000 T0000					
PITCH COMP				00123 N0000			
NO	X	Y	Z	NO	X	Y	Z
000	12	0	30	012	0	0	0
001	0	0	0	013	0	0	0
002	0	0	0	014	0	0	0
003	0	0	0	015	0	0	0
004	0	0	0	016	0	0	0
005	0	0	0	017	0	0	0
006	0	0	0	018	0	0	0
007	0	0	0	019	0	0	0
008	0	0	0	020	0	0	0
009	0	0	0	021	0	0	0
010	0	0	0	022	0	0	0
011	0	0	0	023	0	0	0
NO. 000 =				RUN TIME: 11:26:16			
BITPAR		NUMPAR		PITCH COM		USER PARA	

■ Data recovery and backup

The user data (such as bit parameter, data parameter, and screw-pitch parameter) can be backup (saved) and restored (read) in this GH1000MC system. It doesn't affect the part programs stored in the CNC system while backuping and restoring these data. The backup window is shown as follows:

MDI L		S0000 T0000											
DATA OPERATION				00123 N0000									
[1]RESTORE		[2]BACKUP											
Initial Value reduction run													
☐ PARA (TEST) ☐ PARA (STEP) ☐ PARA (SERVO) ☐ TOOL COMP ☐ PITCH													
RESTORE (C TO CNC) (MT LEVEL)													
☐ PARA ☐ TOOL ☐ PITCH ☐ CURRENT LAD													
RESTORE (U TO CNC) (MT LEVEL)													
☐ PARA ☐ TOOL ☐ PITCH ☐ CURRENT LAD ☐ All Ladder													
HELP: [1]:RESTORE PAGE [2]:BACKUP PAGE													
[IN]:SELECT / CANCEL [OUT]:ALL OPERATION													
PROMPT:													
		RUN TIME: 11:26:56											
Password		Backup		Coordinat		Time							
						Switch							

1. Turn on the parameter switch;

2. Press  key to enter the MDI mode, then press  key ( or  key if necessary) to enter OPERATE DATA window;

■ Press  key to enter Backup DATA Window, press  to enter RECOVERY DATA Window

■ Move cursor to operated option, press  key to select / cancel operation option

Note 1: Don't cut off the power in the backup and restore operation of the data, and no other operation is suggested to be performed before the operation is prompted to be finished.

Note 2: The user above the 3 password level can perform the backup and restore operation of the bit parameter, data parameter and the screw-pitch parameter.

9.3 Password setting and alteration

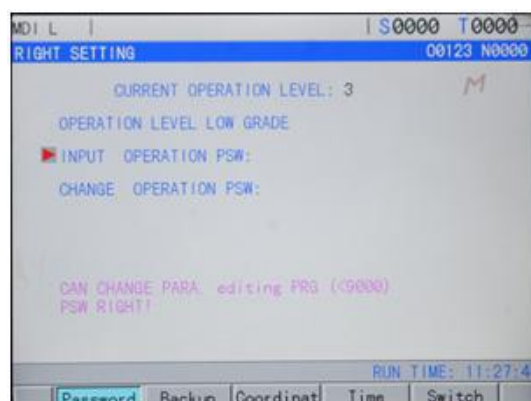
To protect the part programs, CNC parameters from malignant alteration, this GH1000MC provides password setting function that is graded for 4 levels. By descending sequence, they are machine builder (2) level, equipment management (3) level, technician (4) level, machining operation (5) level. The current password level of the CNC system is displayed for "CURRENT LEVEL:_" in the PASSWORD SETTING window.

2 level: the CNC bit parameter, data parameter, screw-pitch parameter, tool offset data, part program edit, PLC ladder transmission etc. are allowed;

3 level: the initial password is 12345, the CNC bit parameter, data parameter, tool offset data, part program edit operations are allowed;

4 level: the initial password is 1234, tool offset data (for tool setting), macro variables, part program edit operations are allowed; but the CNC bit parameter, data parameter, screw-pitch parameter operations are forbidden.

5 level: no password. Only the machine panel operation is allowed, and the operations of part program edit and selection, the alteration operations of CNC bit parameter, data parameter, screw-pitch parameter, tool offset data are forbidden.



After entering the PASSWORD SETTING window, the cursor locates at the "INPUT PASSWORD:" row. It may press the  or  key to move the cursor to the corresponding item.

a) Press  key once, the cursor shifts a row upward. If the current cursor locates at the

“SET LOWER LEVEL” row (1st row), press  key, the cursor shifts to the “ALTER PASSWORD:” row(end row);

- b) Press  key once, the cursor shifts a row downward. If the current cursor locates at the end row, by pressing  key once, the cursor shifts to the 1st row.

9.3.1 Operation level entry

- Move the cursor to the “INPUT PASSWORD:” row after the system enters the PASSWORD SETTING window;
- Input the password (an “*” sign added each time inputting a character);

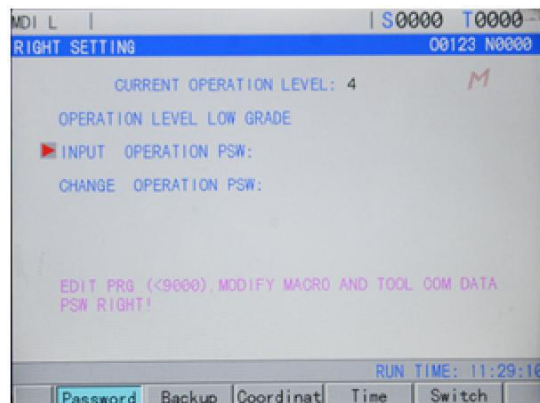
3 Press key  finish the inputting, and the system enters the corresponding password level.

Note: The length of GH1000MC password corresponds to the operation level, which can't be or reduced by user at will. The detailed is as follows:

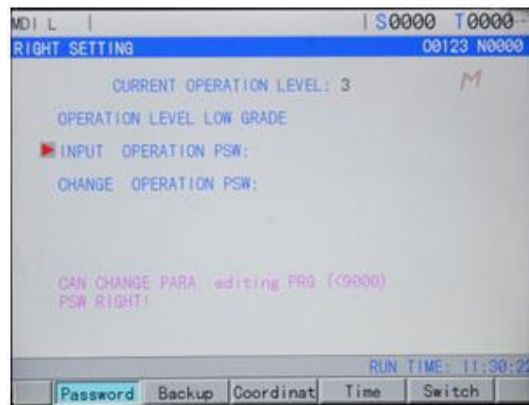
Operation level	Password length	Initial password
3	5 bytes	12345
4	4 bytes	1234
5	No	No

Example:

The current CNC operation level is 4 level, as the following window shows, the 3 level password of CNC is 12345, please alter the current level to the 3 level.



Move the cursor to the “INPUT PASSWORD:” row, key in 12345, then press the key , the CNC prompts “Modify parameter and edit program”, “PASSWORD PASSED.”, and the current level is the 3 level. The display is as follows:



Note: When current operation level is lower than or equal to the 3 level (3, 4, 5 level), this level is not changed if the CNC system is turned on again. If previous level is the 2 level, it defaults the 3 level when the system is turned on again.

9.3.2 Altering the password

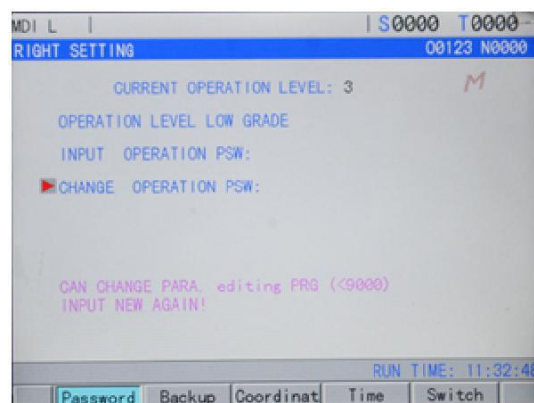
Steps for password alteration:

1 After entering the PASSWORD SETTING window, enter the password by the methods in Section 10.3.1;

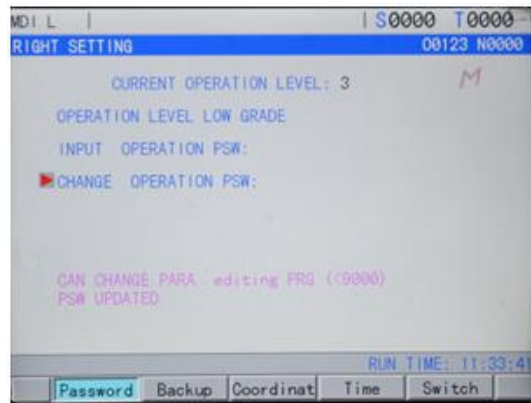
2 Move the cursor to the “ALTER PASSWORD:” row;

3 Key in the new password, then press  key;

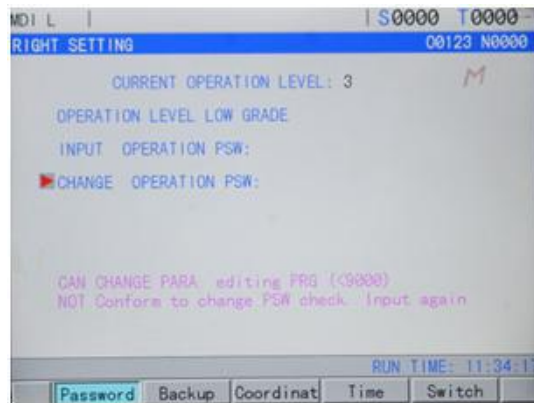
4 The CNC system prompts “PLEASE INPUT USER PASSWORD AGAIN!”, the window display is as follows:



5 After re-inputting the password, press  if the passwords input are identical, CNC prompts “PASSWORD UPDATED.”. So the password alteration is successful.



6 If the inputs of the passwords are not identical, CNC prompts “PASSWORD CHECKOUT ERROR.”, the window is as follows:



9.3.3 Setting the lower password level

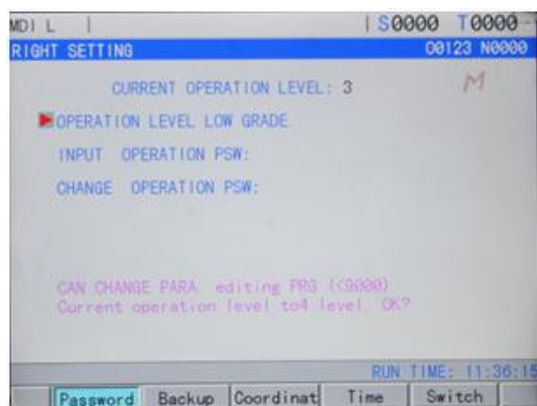
The demotion of the operation level is used to enter a lower level from a higher level, the steps are as follows:

1 After entering the PASSWORD SETTING window, key in the password by the method in Section 9.3.1;

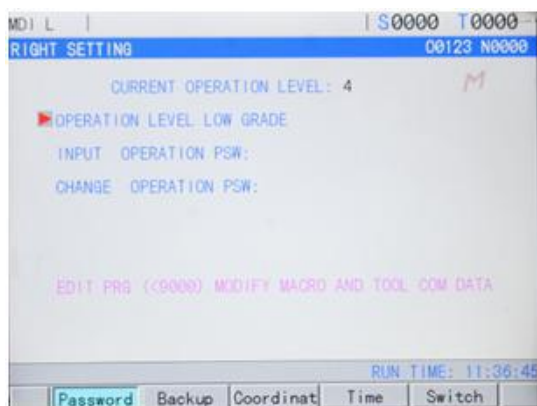
2 Move the cursor to the “SET LOWER LEVEL:” row, if the current CNC operation is the 3 level, the window is as follows:



3 Press  key, the CNC system prompts “CURRENT LEVEL TO 4, MAKE SURE? ”, the window is as follows:



4 Press  key again, if the demotion is successful, the window is as follows:



Note: If the current level is the 5 level, the demotion operation is forbidden.

CHAPTER 10 U DISK OPERATION FUNCTION

10.1 File catalog window

In non-edit mode, press **PROGRAM** to enter the program window, press  to enter [File catalog] window, press **CHANGE** to identify it after U disk is inserted as follows:



The left displays CNC catalog information and the right displays USB disc catalog information. When the system has not checked the U disc, the right does not display the content. The bottom displays the file capacity and user operation prompt. The system only displays “.CNC”, “.NC” and “.txt” in the current file and other extension names are not displayed.

Press **CHANGE** and the cursor is switched from CNC to USB, press  or  to move it.

10.2 File copy

Move the cursor the required CNC format file , press **DATA OUTPUT** to copy.

Copy file from U disk to system: Press **CHANGE** key to switch U disk catalog, and then press key to copy