User Manual

Macro of CNC System

V2.4

Shenzhen Guanhong Automation CO.,LTD
Website: www.szghauto.com
Add:Add: Room 102 QingShuiWan Building,No.7-1 Tangkeng Road, Liuyue community, Henggang Street ,Longgang District,Shenzhen City,Guangdong Province, China
Post code: 518110


SZGH

-Catalog-

Chapter 1 G MACRO

1.1 Non-Mode Macro Command G65

Format: G65 P_ L_ A_ B_ C_ .....

Non-mode macro command G65 only work at current line , which is different to mode macro command(G66),which always work until macro cancel command(G67)

P_ : Specify name of macro program, E.g: P6000 , name of specified macro program is 6000 .

L_: Set times of call macro program

<A_B_C_... ...> : Argument , which is used for transfer data to macro variable (#**) , Transferring table is as following

Argument	Variable	Argument	Variable	Argument	Variable
A	#0	I	#7	T	#14
B	#1	J	#8	U	#15
C	#2	K	#9	V	#16
D	#3	M	#10	W	#17
E	#4	Q	#11	K	#18
F	#5	R	#12	Y	#19
H	#6	S	#13	Z	#20

Warning:

1. Macro variables #100-#155&#190-#201 was occupied by system, user cannot use.

2. User cannot use G70,G71,G72,G73,G92,G76 etc loop command on Macro program.

Note: the address G, L, N, Q, P can't be used as user-defined variables.

Example: Main Program: 9000

```
G00 X0 Z0
G65 P8000 L1 A5 B6
G0 X0 Z0
M30
```

Macro Program: 8000

```
N1 #2=#0+#1
N2 IF(#2 EQ 10) GOTO 4
N3 G00 X#2
N4 G00 Z#1
N5 M99      ; Return
```

1.2 Mode Macro Command G66/G67

G66 is mode macro command , G67 is cancel mode macro command

Format: G66 P_ L_ A_ B_ C_ .....

G67

G66 Mode macro command,which always call macro program until macro cancel command(G67)

P_ : Specify name of macro program, E.g: P7000 , name of specified macro program is 7000 .

L_: Set times of call macro program

<A_B_C_... ...> : Argument , which is used for transfer data to macro variable (#**) , the transferring table is same as above table.

Example:

```
Main Program : 4000
G00 X0 Z0
G66 P6000 L2 A5 B6
A8 B1
A9 B10
G67
M30
Macro Program: 6000
N1 #2=#0+#1
N2 IF (#2 EQ 10) GOTO 4
N3 G00 X#2
N4 G00 Z#1
N5 M99              ; Return
```

1.3 User-defined G Macro Program

Program Names of G101-G170 , user-defined G codes, are corresponding to ProgramG101 - ProgramG170; G101~G150 are non-model codes,

G Macro program(ProgramG101 - ProgramG170) must be edited on computer, and then copy to U-disk , and restore into cnc system.

1.4 Macro program instruction

1.4.1 Input Instruction: WAT

Waiting for the input port X valid or invalid instruction

Format: WAT+ (-) X(Y/M)**+(-)X**+(-)X**+(-)X**+(-)X**

Attention: "+" to means wait for the input is effective;

"-" means wait for the input is invalid;

CNC only can detect only one Y or M , X is without limit.

"X" means the input port X00-X55; see the I/O diagnosis;

1.4.2 Output Instruction: OUT

Set the output port Y is valid or invalid instruction

Format: OUT +(-)Y

Attention: "+" means the output is effective;

"-" means the output is invalid;

"Y" means the output port Y00-Y31; see the I/O diagnosis;

1.4.3 Assignment Instruction: =

Explanation: used for assignment of a variable

Eg.: #251=890.34 #450=#123

And also it could be mathematical expression , example.: #440=#234+#470

1.4.4 Unconditional Jump: GOTO n

"GOTO n" is the command that for jump to the program line that is specified by sequence number (N**) unconditionally. n is the sequence number.

E.g.: GOTO 5 ; // Jump to N5 program line.

Note: when specified program line , n , is beyond sequence number of N1-N99999, cnc system will hint error.

n , program line,could be macro variable (#**)

E.g.: GOTO #100

1.4.5 Conditional Jump

1) IF (Conditional express) GOTO n

If condition is met, execute GOTO n ,jump to N** program line; if the condition isn't met, execute the next segment.

Example: N1 IF(#200 EQ 1) GOTO 20
N10 G00 X0
N20 G00 Z0

Explanation: If #200 is equal to 1, system will execute GOTO 20 , jump to N20 , and execute "G00 Z0", if #200 isn't equal to 1, system don't execute operation of "GOTO 20" ,and will execute next segments , "G00 X0",and then execute "G00 Z0".

2) IF (Conditional express) THEN <A Expression>
<B operational segment>

If condition is met, system execute A expression , and then execute B operational segment ; if condition is not meet, execute the next segment , B operation.

Example: #101=0
N1 IF(#100 EQ 1) THEN #101=1
N2 IF(#101 EQ 1) GOTO 4
N3 G00 X100
N4 G00 Z100

Explanation: If #100 is equal to 1, system will execute "#100=1", and then judge #101 is equal to 1 , jump to N4 & "execute G00 Z100" ; if #100 isn't equal to 1, system will judge #101 also isn't equal to 1 directly , and execute "G00 X100" & "G00 Z100".

NOTE: 1.<A expression> normally is assignment statement.

2. <A expression> after THEN must exist, otherwise system will hint grammatical errors.

Prolongation:

3) IF(conditional express)
<A operational command>
ELSE
<B operational command>
ENDIF

4) IF(conditional express)
<A operational command>
ELIF
<B operational command>
ENDIF

1.4.6 Loop Command

Format: (Conditions Initialization)
WHILE (conditional expression) DO n
<A operational segments>
[Alter condition of loop]
END n
<B operational segments>

When conditions are met during WHILE cycle command, execute the operational segments between DO n and END n . Otherwise,when condition isn't met, jump to the program line after END n ,also execute B operational segments.

We can nest for loops by placing one loop within another.

Note: 1.There must have operational codes that are for change condition at operational segments ,which is between Do n & END n. Otherwise system will enter endless loop.

2.Nesting of macro program loop statements of SZGH CNC system is 3 pcs of loops at most . Also n only could be 1 , 2 , 3 .

3.n of "DO n" & "END n" must keep same.

Example:
```
#100=2 #150=5 #200=25
WHILE （#100 LT 3）DO 1
 G00 X100
 WHILE （#150 EQ 5）DO 2
 G00 Y100
 WHILE （#200 GE 20）DO 3
 G00 Z100
 #200=#200-2
 END 3
 #150=#150-1
 END 2
 #100=#100-1
 END 1
```

1.5 Operators' meaning

Operator	Sign	Ex.	Operator	Sign	Ex.	Operator	Sign	Ex.
EQ	=	equal	GT	>	greater	LT	<	Less
NE	≠	unequal	GE	≥	G&E	LE	≤	L&E

1.6 Arithmetic & Logic Operation

Table:

Function	Format	Attention
Definition	#i = #j	
Addition Subtraction Multiplication Division	#i = #j + #k ; #i = #j - #k ; #i = #j * #k ; #i = #j / #k ;	
Sin Asin	#i = SIN(#j) ; #i = ASIN(#j);	90.5 degrees means 90 degrees & 30 points

Cos Acos Tan Atan	#i = COS(#j) ; #i = ACOS(#j); #i = TAN(#j); #i = ATAN(#j);	
Square root Absolute value Rounding off Round down Round up Natural logarithm Exponential function	#i = SQRT(#j); #i = ABS(#j) ; #i= ROUND(#j); #i = FIX(#j); #i = FUP(#j); #i = LN(#j); #i = EXP(#j);	
Or Exclusive or And	#i = #j OR #k ; #i = #j XOR #k ; #i = #j AND #k ;	Executing with binary system

1.7 Local Variable

#0--#20 : local variables only can be used to store data in macro program, such as a result of operation, when power is off, the local variables are initialized to the empty. The argument assignment to the local variable when calling the macro program.

1.8 Global Variable

#21--#600 : Their meanings are the same in different macro program.

When power is off, the variable #21--#100 is initialized to zero, the variable #101--#600 data is saved not to loss even if the power is off.

1.9 System Variable

#1000-- : the system variables are used to change various data when reading the running CNC. For example, the current position and the compensation of tool.

Special Attention: macro variables #100--#155 and #190--#202 have been used by the system, users can not use.

1.10 System Parameter Variable

#1001--#1099 : Value of X-axis length compensation for T1--T99(Unit: um)
#1101--#1199 : Value of D1 radius compensation for T1--T99(Unit: um)
#1201--#1299 : Value of Y(C)-axis length compensation for T1--T99(Unit: um)
#1301--#1399 : Value of D2 radius compensation for T1--T99(Unit: um)
#1401--#1499 : Value of Z-axis length compensation for T1--T99(Unit: um)
#1501--#1599 : Value of D3 radius compensation for T1--T99(Unit: um)
#1601--#1699 : Value of A-axis length compensation for T1--T99(Unit: um)
#1701--#1799 : Value of D4 radius compensation for T1--T99(Unit: um)

1.11 I/O variable

#1800: X00-X07 (D0-D7) ; input resistor
#1801: X08-X15 (D0-D7) ; input resistor

#1802: X16-X23 (D0-D7) ; input resistor
#1802: X16-X23 (D0-D7) ; input resistor
#1803: X24-X31 (D0-D70 ; input resistor
#1804: X32-X39 (D0-D7) ; input resistor
#1805: X40-X47 (D0-D7) ; input resistor
#1806: X60-X67 (D0-D7) ; input resistor
#1807: X74-X81 (D0-D7) ; Alarm of driver/Spindle
#1808: Y00-Y15 (D0-D15) ; output resistor
#1809: Y16-Y31 (D0-D15) ; output resistor
#1810: Y32-Y47 (D0-D15) ; output resistor

Warning: 1. Macro variables #100-#155&#190-#201 was occupied by system, user cannot use.

2. User cannot use G70,G71,G72,G73,G92,G76 etc loop command on Macro program.

Note: the address G, L, N, Q, P can't be used as user-defined variables.

1.12 Message Hint Dialog Box

Format: MSG(hint words) or MSG[hint words] ;

Hint words is that user want to hint message on cnc system.

Note: 1. This code can be used on normal NC programs.

2. After hint message, cnc system will pause program automatically.

Format: STAF(hint words) or STAF[hint words];

Hints words is that user want to hint message on cnc system. And CNC system don't pause program automatically.

1.13 Build Processing Program Automatically

1.13.1 New/Open a program

Format: FILEON(Program) or FILEON[Program]

Example: FILEON(AABBCC) or FILEON[AABBCC]

It means that new or open a program "AABBCC"

1.13.2 Close program

Format: FILECE

It means that close current opening program, if without this code, system will close current program after program is finished.

1.13.3 Write codes into program

Format: FILEWD(Blocks) or FILEWD[Blocks]

Example: FILEWD(G54G0X0Z0) or FILEWD[G54G0X0Z0]

It means that write a blocks of "G54G0X0Z0" into current opening program.

1.13.4 Write current absolute Coordinate into program

Format: FILEWC

It means that write current absolute Coordinate value into program.

Example:

```
G0X0Z0
FILEON[AABBCC]
FILEWD [G54G0X0Z0]
G1X45Z89
```

After finished this program, system will new a program of "AABBCC" under directory of program, its blocks is :

G54G0X0Z0

X45Z89

X99Z76

FILEWC
G1X99Z76
FILEWC
FILECE

1.13.5 Save all global variables with program format

FILEMS(ABC) or FILEMS(ABC)

Save #21~#999 to program of "ABC"

If user want to read these variables, it just needs to execute this "ABC" program, or take this program as subprogram.

1.13.6 Application Example

G0 X100 Z2
FILEON(SCANNER01)
G22 L10
G31 Z-10 P1001 F100
FILEWC
G0 Z2
U-1
G800
This is example program for scanning,which will generate program: SCANNER01

Chapter 2 M MACRO

2.1 User-defined M Macro Program(M881-M889)

M881-M889 , user-defined M macro instruction ,whose corresponding macro program is ProgramUser1 - ProgramUser9. And related M codes can call corresponding macro program. When we use M codes in processing program, CNC system will executive inner codes of M macro program , which is similar to subprogram.

M codes	Macro Program	M codes	Macro Program
M881	ProgramUser1	M886	ProgramUser6
M882	ProgramUser2	M887	ProgramUser7
M883	ProgramUser3	M888	ProgramUser8
M884	ProgramUser4	M889	ProgramUser9
M885	ProgramUser5	...	...

ProgramTool,ProgramM6 are edited by these basic codes.

M Macro program(ProgramUser1 - ProgramUser9) must be edited on computer, and then copy to U-disk , and restore into cnc system.

2.2 Instructions of M Macro Program

2.2.1 Output Instruction:

OUT+(-/A/P/R)Y(M)**+(-/A/P/R)Y(M)**+(-/A/P/R)Y(M)**+(-/A/P/R)Y*(M)**

Explanation: Set output Y, M is valid or invalid

"+" means the output is valid ;

"-" means the output is invalid;

“A” table input inversion（valid becomes invalid, invalid becomes valid);

“P” after judged nearby for turret, if forward rotation, output is valid; if reverse rotation, output is invalid;

“R” after judged nearby for turret, if forward rotation, output is invalid, if reverse rotation, output is valid;

Eg. : OUT+Y3:set output of Y3 is valid ; OUT-M28:set output of M28 is invalid.

OUT+Y5-Y7+Y9+Y11-Y15+M300 : set output of Y5, Y9, Y11 & M300 is valid, output of Y7 & Y15 is invalid.

OUTPY18RY19; when forward rotation of turret,CNC will activate output Y18, when reverse rotation of turret, it will activate output Y19.

2.2.2 Output Instruction: WHEN**OUT

Used to output signal after judging tool number, format:

WHEN**OUT+(-/A/P/R)Y(M)**+(-/A/P/R)Y(M)**(-/A/P/R)Y(M)**+(-/A/P/R)Y(M)**

Explanation: “ ** ” of “ WHEN**OUT ” means tool number

“+” means the output is valid ;

“-” means the output is invalid;

“A” table output inversion（valid becomes invalid, invalid becomes valid);

“P” after the tool post is judged nearby, if the forward rotation output is valid, if it is reverse rotation, the output is invalid;

“R” after the tool post is judged nearby, if the forward rotation output is invalid, if it is reverse rotation, the output is valid;

Eg:WHEN7OUT+Y5-Y7+Y9+Y11-Y15

Means: When changing the 7 tools, set output of Y5, Y9 & Y11 is valid, output of Y7 & Y15 is invalid.

Eg:WHEN8OUT+M12-M13+Y14+Y8-Y16

Means: When changing the 8 tools, set output of M12, Y14 & Y8 is valid, output of M13 & Y16 is invalid.

2.2.3 Wait Instruction: WAT+/- X/Y/M

Explanation:Waiting for X/Y/M is valid or invalid; when the conditions are met, macro program can executive following instructions.

Format: WAT+(-)X(Y/M)**+(-)X**+(-)X**+(-)X**+(-)X**+(-)X**

Explanation: “+” means the wait is valid ;

“-” means the wait is invalid;

Y or M must have at most one or none, pieces of X can be any ;

Eg.: WAT+X01-X02 : Wait X01 is valid,X02 is invalid,and then run following codes.

2.2.3.1) Max waiting time of wait Instruction

Used to limit max waiting time for waiting for the instruction WAT.

Format: MAXWAT****

Explanation: Number range after instruction is 0-99999, unit is ms.

After the time expires, the system will alarm and exit the program.

Assigning a value of 0 or the MAXWAT command table alone disables this function, there is no need to limit the longest waiting time for waiting for the instruction WAT.

2.2.3.2) Wait for condition of instruction WAT to meet holding time: HOLDWAT

Used to specify the hold time when the WAT condition of the waiting instruction is satisfied.

Format: HOLDWAT****

Explanation: Number range after instruction is 0-99999, unit is ms.

Assigning a value of 0 or the MAXWAT command table alone disables this function, there is no need to limit the longest waiting time for waiting for the instruction WAT.

2.2.3.3) Mode of max waiting for instruction WAT: MODWAT

This instruction is used in conjunction with the instruction MAXWAT

Format:MODWAT1 (Default) /MODWAT2/MODWAT3/MODWAT4

MODWAT1 :After waiting for the instruction WAT time to exceed, the system will dialog box alarm to exit.

MODWAT2 :After waiting for the instruction WAT time to exceed, the system will dialog box alarm to continue the program running.

MODWAT3 After waiting for the instruction WAT time to exceed, the system will not dialog box alarm to exit.

MODWAT4 After waiting for the instruction WAT time to exceed, the system will not dialog box alarm to continue the program running.

Eg: If X0 signal is not received within 5 seconds, alarm 1 will be reported.

MODWAT4

MAXWAT5000

WAT+X0

IF(-X0) THEN

OUT+M81

ERREXIT

ENDIF

2.2.4 Delay Instruction: PAUS

Explanation: Instruction that used for delaying(0-99999ms), unit: ms ;

Eg.: PAUS100 : delay 100 ms .

PAUS1000 : delay 1 second,also 1000ms.

2.2.5 Suspend Instruction:M36

Explanation: used for single stage tool change, the tool change program will be suspended until it is valid again

2.2.6 Assignment Instruction: =

Explanation: used for assignment of a variable

Eg.: #251=890.34 #450=#123

And also it could be mathematical expression , eg.: #440=#234+#470

2.2.7 Refresh tool display Instruction

Used to refresh the contents of the "Tools" column on the main page of the system.

Format: DISP

2.2.8 Save tool number Instruction: SAETOOL

It is used to keep the tool number when the tool number is not determined by the external switch. Usually used at the end of the entire program.

Format: SAVETOOL

2.2.9 Waiting for tool in position Instruction: CASE

Wait for a certain tool to change the tool in place, and judge the conditions pointed out by the instruction, when the conditions are met, the tool change is considered to be in place, and the program is executed again.

Format CASET**

Explanation: T**: Tool number ;

“+” means input is valid, “-”means input is invalid,

“X” means address of input points.

CASET1-X16+X17-X18-X19+X20

Example: Wait for the 1 tool to rotate to the position, when the 1 is in place, the X16, X18, X19 input signals are invalid, and the X17, X20 input signals are invalid.

2.2.10 Information Prompt Dialog

Format: MESSAGEBOX(InfoPrt1) , “ InfoPrt1” is the prompt information.

Or Shorthand: MSG(parameter 1); “parameter 1” is information string.

2.2.11 Information prompt of current tool change status

Format:STATUSINF0(parameter 1) ; parameter 1 is information string.

Or Shorthand: STAF(parameter 1); parameter 1 is information string.

Note: When only STATUSINF0 or STAF or STATUSINF0() or STAF() is edited, it means that the current information prompt is closed.

2.2.12 Tool selection

Format: CHOOSET(parameter 1,parameter 2);//parameter 1 is 1 means tool selection in umbrella type tool magazine, parameter 2 is tool number.

TS0 : current tool number on spindle, TAIM : tool code number

2.2.13 Conditional Statement

Conditional statement: when condition is met, executive codes after THEN, otherwise jump conditional codes.

Format: IF(**) THEN

ENDIF

“ ** ” :auxiliary relays(M), Input points(X) , Output points(Y) or macro variable(#*) .

+M** : When auxiliary relay is valid, condition is met .

-M** : When auxiliary relay is invalid, condition is met .

+X** : When input point X is valid, condition is met .

-X** : When input point X is invalid, condition is met

+Y** : When output point Y is valid, condition is met .

-Y** : When output point Y is invalid, condition is met .

+#** : When macro variable is 1(Non 0), condition is met .

-#** : When macro variable is 0, condition is met .

Eg.1 : When input X13 is valid , prompt dialog and exit.

```
IF(+X13) THEN
MESSAGEBOX(ERROR:CAN'T EXCHANGE TOOL!)
RETURN
ENDIF
```

Eg.2 : When input Y13 is valid , prompt dialog and exit.

```
IF(+Y13) THEN
MESSAGEBOX(ERROR:CAN'T EXCHANGE TOOL!)
RETURN;
ENDIF
```

E.g.3: When macro variable #313 is 1(non-0) , pop up dialog and exit.

```
IF(+#313)THEN
MESSAGEBOX(ERROR: COOLANT PUMP IS OVERLOAD!)
RETURN
ENDIF
```

Conditional sentences are added on the basis of the same appeal function:

1> IF [Logical expression]THEN

……

……

ENDIF

2> IF [Logical expression] THEN #345=235 (Assignment statement)

3> IF [Logical expression] THEN OUT+Yxx (Jump directly out of the output port statement)

4> IF [Logical expression] GOTO xxx (Jump statement)

Note: [Logical expression] The format is the same as the user macro program format, with GT/NE/LT and other logical judgments and mathematical calculation expressions available.

2.2.14 Move Coordinate Axes

Format: MOVE(G_ , F_ , X_ ,Y_, Z_, A_, B_, W_) ;

1st Parameter: G_ , which could be G90 or G91 , G90: absolute programming ; G91: Incremental programming.Value of axes is absolute or incremental ;

2nd Parameter: F_ , which is set for feeding speed ;

3rd Parameter: X_,Y_,Z_,A_,B_, specify coordinate value of each axes ;

4th Parameter: W_ , which is set for input point ,which is met condition, to stop moving. Eg.: W+5, when input point X5 is valid,stop moving .

E.g.: MOVE(G90,F8000,X-100) ; //X axis moves to X-100 of machine coordinate

MOVE(G91,F8000,Z-100) ; //Z axis moves -100mm

MOVE(G91,F8000,Y-300,W-8); //Y axis moves -300mm,when input point 8 is invalid,stop moving.

2nd & 3rd parameter can be specified by macro variable(#**)

Eg.: MOVE(G91,F#231,Y#240,W+2);//Y axis move #240 with #231 speed ; when input point X2 is valid, stop moving;

Suggestion: This codes could be used to tool setting and sets of tool compenstion with tool setting gauge.

2.2.15 Set & Save Current Workpiece Coordinate

Format: SETWK(X_ , Y_ , Z_ , A_ , B_) ;

Eg.: SETWK(X23.45,Z567.89); // set X-axis to 23.45 , set Y-axis to 567.89 at current coordinate system.

Value of XYZAB axis also can be specified by macro variable(#**) ;

Eg.: SETWK(X#238,Z#237); // at current coordinate system ,set value of X-axis to #238 , set value of Z-axis to #237.

2.2.16 Set & Save Current Machine Coordinate(G53)

Format : SETMH(X_ , Y_ , Z_ , A_, B_)

Eg.: SETMH(X23.56,Z567.89);// Set X-axis value to 23.56 , set Z-axis value to 567.89 at current machine tool coordinate ;

Value of parameters X_/Y_/Z_/A_/B_ can be specified by macro variable(#**).

E.g.: SETMH(X#238,Z#239) ; // Set X-axis value to #238 , set Z-axis value to #239 at current machine tool coordinate ;

2.2.17 Set&Save offset of current workpiece coordinate system

Format : SETMH(parameter X , parameter Y , parameter Z , parameter A, parameter B)

Eg: SETMH(X23.56,Z567.89);// Set X-axis value to 23.56 , set Z-axis value to 567.89 at current machine tool coordinate ;

Value of parameters XYZAB can be specified by macro variable (#**).

E.g.: SETMH(X#238,Z#237) ; // Set X-axis value to #238 , set Z-axis value to #237 at current machine tool coordinate ;

2.2.18 Count Instruction: COUN

Wait for the instruction of a certain tool to change the position through a tool counting signal, and judge the specified tool positioning signal condition. When the condition is satisfied, the tool change is considered to be in place, and the program is executed again.

Format: COUNT**+(-)X**+(-)X**

Explanation: “T**” table tool number, TT table instruction tool number passed.

The first input point X exponential knife signal, ‘+’ means the number of dollars on the rising edge, and ‘-’ means the number of dollars on the falling edge.

The second input point X index knife signal, ‘+’ means valid positioning, and ‘-’ means invalid positioning.

Two input points X are the same point. There can be no positioning signal for the second input point.

According to the setting of the parameter "unidirectional / bidirectional tool selection", the increase or decrease of the tool number is determined. If it is unidirectional, the tool number increases. If it is bidirectional, the increase or decrease of the tool number is determined by the principle of nearest tool selection.

The parameter "Whether the tool number is determined by the input point" is changed to not determined by the input point.

Eg: COUNTT+X11-X11 Select the tool number transmitted in the T command, the tool counting signal is X11, the rising edge counting tool, the positioning signal is X11, and the

positioning is invalid.

COUNTT+X10 Select the tool number transferred in the T command, the tool counting signal is X10, and the rising edge counts the tool without detecting the positioning signal.

COUNT6+X10-X10 Select the 6 tool, the tool counting signal is X10, the rising edge counting tool, the positioning signal is X10, and the invalid means the positioning signal.

COUNT7+X6+X8 Select the 7 tool, the tool counting signal is X6, the rising edge counting tool, the positioning signal is X8, and the valid means the positioning signal.

Note: The tool counting instruction can only be used for the tool magazine with low speed rotation.

2.2.19 Digital Tool output instruction: CUNOUT

After outputting the tool magazine rotation command, it will automatically close the tool magazine forward and reverse output points set in the parameters after waiting for the tool change to be completed.

Format: CUNOUT+(-/A/P/R)Y**+(-/A/P/R)Y**+(-/A/P/R)Y**

Explanation:

"+" means the output is valid ; "-" means the output is invalid;

"A" table output inversion (valid becomes invalid, invalid becomes valid);

"P" after the tool post is judged nearby, if the forward rotation output is valid, if it is reverse rotation, the output is invalid;

"R" after the tool post is judged nearby, if the forward rotation output is invalid, if it is reverse rotation, the output is valid;

Eg:CUNOUT+Y9PY5RY7+Y11-Y15

The parameters of the machining center need to be set as follows:

P32, tool selection method/count signal (bit0:0 means unidirectional/1 means two-way tool selection;2 bits and 4 bits is digital knife filter setting)

P25, Tool magazine count signal [Rising edge "1000+number", Falling edge "2000+number"]

P26, tool magazine positioning signal[1000+number]

P27, tool magazine forward output point[1000+number]

P28, tool magazine reverse output point[1000+number]

Note: this tool counting instruction can be used for high-speed rotating tool magazines.

2.2.20 Jump statement: G0T0

Format: G0T0 xx

Example: OUT-Y8

N12; The marked line should occupy one alone line

OUT+Y4+Y6

PAUS5000

WAT+X23

G0T0 12; Jump to N12 marked line

2.2.21 Save macro variables to memory: SAVEMACR

Format: SAVEMACR

2.2.22 Error & Exit

Format: ERREXIT

When macro program have errors, it will exit macro program

2.2.23 Return

Format: RETURN

When macro program is processed successfully,and use this code to exit macro program and return back.

2.2.24 Special variable interpretation

CURTS: Current tool pocket number.

CURTH: Current tool length compensation number.

TAIM: The value passed by the T instruction.

TS(xx): Tool number in tool holder xx.

TS(0) or TS0: Tool number on the spindle.

2.2.25 Pause instruction:M00 or M0

It is used for ordinary control program, it is that the control program will be paused when it runs, and it will continue to run after pressing the run key.

Format:M00 or M0

2.2.26 Set&Save current tool length compensation value

Format: SETTH(parameter X, parameter Y, parameter Z, parameter A, parameter B);

Eg: SETH(X23.56, Z567.89);//Setting current tool X coordinate is 23.56 mm and Z coordinate is 567.89 mm under length compensation.

Parameter XYZAB can use macro # specify number.

Eg: SETTH(X#238,Z#237);//Setting current tool X coordinate is #238 mm, Z coordinate is #237 mm.

2.2.27 Set&Save current tool radius compensation value

Format: SETTD(parameter);

In development……

2.2.28 Instructions on file operations

2.2.28.1) Create open file instructions

FILEON(parameter) or FILEON[parameter]

Eg:

FILEON(AABBCC) or FILEON[AABBCC]

Table create open a file AABBCC

2.2.28.2) Close file instruction

FILECE table close current open file, if there is not instruction, the program will auto close current open file after running.

2.2.28.3)Instructions to write a string of characters to an open file

FILEWD(parameter) or FILEWD[parameter]

Eg:

FILEWD(G54G0X0Z0) or FILEWD[G54G0X0Z0]

Table write string G54G0X0Y0 into the open file

2.2.28.4)Write absolute coordinate instruction of current feed axis to the opened file:

FILEWC

2.2.29 Control instruction for serial communication

2.2.29.1) Configure the serial port parity mode

Format:COMP1 table odd parity

COMP2 table even parity

COMP3 table no parity

2.2.29.2) Clear the serial port data receiving buffer

Format: COMC table clears the serial port receive buffer.

2.2.29.3)Send a character from the serial port

Format: COMS(parameter); parameter is a string or macro(the value of macro is ASCII code corresponding to the character)

Eg: COMS(A); Table send string A out.

COMS(#560); Table ASCII code corresponding to the macro #560 send out.

2.2.29.4)Send a character from the serial port

Format: COMR(parameter); The parameter is a macro variable, and the macro variable saves the character value of the receiving tool.(The value of the macro variable is the ASCII code corresponding to the character.)

Eg: COMS(#561); Table assign the ASCII code value corresponding to the received character to the macro variable #561.

Chapter 3 Variable

3.1 Local Variable

#0--#20 : local variables only can be used to store data in macro program, such as a result of operation, when power is off, the local variables are initialized to the empty. The argument assignment to the local variable when calling the macro program.

3.2 Global Variable

#20--#600 : Their meanings are the same in different macro program.

When power is off, the variable #20--#100 is initialized to zero, the variable #100--#600 data is saved not to loss even if the power is off.

3.3 System Variable

#1000-- : the system variables are used to change various data when reading the running CNC. For example, the current position and the compensation of tool.

Special Attention: macro variables #100--#155 and #190--#202 have been used by the system, users can not use.

3.3.1 System Parameter Variable

#1000--#1099 : Value of X-axis length compensation for T0--T99(Unit: um)

#1100--#1199 : Value of D0 radius compensation for T0--T99(Unit: um)

#1200--#1299 : Value of Y(C)-axis length compensation for T0--T99(Unit: um)

#1300--#1399 : Value of D2 radius compensation for T0--T99(Unit: um)

#1400--#1499 : Value of Z-axis length compensation for T0--T99(Unit: um)

#1500--#1599 : Value of D3 radius compensation for T0--T99(Unit: um)

#1600--#1699 : Value of A-axis length compensation for T0--T99(Unit: um)

#1700--#1799 : Value of D4 radius compensation for T0--T99(Unit: um)

3.3.2 I/O variables

#1800: X00-X07 (D0-D7) ; input resistor

#1801: X08-X15 (D0-D7) ; input resistor

#1802: X16-X23 (D0-D7) ; input resistor

#1803: X24-X31 (D0-D70 ; input resistor

#1804: X32-X39 (D0-D7) ; input resistor

#1805: X40-X47 (D0-D7) ; input resistor

#1806: X60-X67 (D0-D7) ; input resistor

#1807: X74-X81 (D0-D7) ; Alarm of driver/Spindle

#1808: Y00-Y15 (D0-D15) ; output resistor

#1809: Y16-Y31 (D0-D15) ; output resistor

#1810: Y32-Y47 (D0-D15) ; output resistor

#1811: M00-M15(D0-D15) ; auxiliary relay

#1812: X101-X102(D0-D11) ; input of operational panel

#1813: Y50-Y57(D0-D7) ; servo enable

#1814: M06-M30(D0-D15) ; auxiliary relay

#1815: M32-M47(D0-D15) ; auxiliary relay

#1816: M48-M63(D0-D15) ; auxiliary relay

#1817: M64-M79(D0-D15) ; auxiliary relay

#1818: M100-M115(D0-D15) ; auxiliary relay

#1819: M80-M95(D0-D15) ; auxiliary relay

#1820: M120-M135(D0-D05) ; auxiliary relay

3.3.3 Parameter Variable

#1850: X-axis Offset value of G54 workpiece coordinate system(unit:mm)

#1851: X-axis Offset value of G55 workpiece coordinate system(unit:mm)

#1852: X-axis Offset value of G56 workpiece coordinate system(unit:mm)

#1853: X-axis Offset value of G57 workpiece coordinate system(unit:mm)

#1854: X-axis Offset value of G58 workpiece coordinate system(unit:mm)

#1855: X-axis Offset value of G59 workpiece coordinate system(unit:mm)

#1856: Y-axis Offset value of G54 workpiece coordinate system(unit:mm)

#1857: Y-axis Offset value of G55 workpiece coordinate system(unit:mm)

#1858: Y-axis Offset value of G56 workpiece coordinate system(unit:mm)

#1859: Y-axis Offset value of G57 workpiece coordinate system(unit:mm)

#1860: Y-axis Offset value of G58 workpiece coordinate system(unit:mm)

#1860: Y-axis Offset value of G59 workpiece coordinate system(unit:mm)

#1862: Z-axis Offset value of G54 workpiece coordinate system(unit:mm)

#1863: Z-axis Offset value of G55 workpiece coordinate system(unit:mm)

#1864: Z-axis Offset value of G56 workpiece coordinate system(unit:mm)

#1865: Z-axis Offset value of G57 workpiece coordinate system(unit:mm)

#1866: Z-axis Offset value of G58 workpiece coordinate system(unit:mm)

#1867: Z-axis Offset value of G59 workpiece coordinate system(unit:mm)

#1868: A-axis Offset value of G54 workpiece coordinate system(unit:mm)

#1869: A-axis Offset value of G55 workpiece coordinate system(unit:mm)

#1870: A-axis Offset value of G56 workpiece coordinate system(unit:mm)

#1870: A-axis Offset value of G57 workpiece coordinate system(unit:mm)

#1872: A-axis Offset value of G58 workpiece coordinate system(unit:mm)

#1873: A-axis Offset value of G59 workpiece coordinate system(unit:mm)

#1874: X-axis Coordinate Value of 1st reference point (unit:mm)

#1875: X-axis Coordinate Value of 2nd reference point (unit:mm)

#1876: X-axis Coordinate Value of 3rd reference point (unit:mm)

#1877: X-axis Coordinate Value of 4th reference point (unit:mm)

#1878: Y-axis Coordinate Value of 0st reference point (unit:mm)

#1879: Y-axis Coordinate Value of 2nd reference point (unit:mm)

#1880: Y-axis Coordinate Value of 3rd reference point (unit:mm)

#1880: Y-axis Coordinate Value of 4th reference point (unit:mm)

#1882: Z-axis Coordinate Value of 0st reference point (unit:mm)

#1883: Z-axis Coordinate Value of 2nd reference point (unit:mm)

#1884: Z-axis Coordinate Value of 3rd reference point (unit:mm)

#1885: Z-axis Coordinate Value of 4th reference point (unit:mm)

#1886: A-axis Coordinate Value of 0st reference point (unit:mm)

#1887: A-axis Coordinate Value of 2nd reference point (unit:mm)

#1888: A-axis Coordinate Value of 3rd reference point (unit:mm)

#1889: A-axis Coordinate Value of 4th reference point (unit:mm)

#1890: Set of Metric/Inch in system

#1890: Name of 4th axis

#1892: Function Set of 4th axis

#1893: Rotary mode when 4th axis is as rotary axis

#1894: Display of relative coordinate when 4th axis is as rotary axis

#1895: Display of workpiece coordinate when 4th axis is as rotary axis

#1896: Reverse Backlash Compensation of X-axis (unit: um)

#1897: Reverse Backlash Compensation of Y-axis (unit: um)

#1898: Reverse Backlash Compensation of Z-axis (unit: um)

#1899: Reverse Backlash Compensation of A(4th)-axis (unit: um)

#1900: Feeding axis use electric gear(0: use , non-0: no)

#1900: Numerator of X-axis's electron gear (0—999999999)

#1902: Denominator of X-axis's electron gear (0—999999999)

#1903: Numerator of Y-axis's electron gear (0—999999999)

#1904: Denominator of Y-axis's electron gear (0—999999999)

#1905: Numerator of Z-axis's electron gear (0—999999999)

#1906: Denominator of Z-axis's electron gear (0—999999999)

#1907: Numerator of A-axis's electron gear (0—999999999)

#1908: Denominator of A-axis's electron gear (0—999999999)

#1909: Spindle use electric gear (0: use , non-0: no)

#1900: Numerator of electron gear ratio of spindle in Low grade (0—999999999)

#1900: Denominator of electron gear ratio of spindle in Low grade (0—999999999)

#1902: Numerator of electron gear ratio of spindle in Low grade (0—999999999)

#1903: Denominator of electron gear ratio of spindle in High grade (0—999999999)

#1904: Direction signal of X-axis(0: reverse , non-0: normal)

#1905: Direction signal of Y-axis(0: reverse , non-0: normal)

#1906: Direction signal of Z-axis(0: reverse , non-0: normal)

#1907: Direction signal of A-axis(0: reverse , non-0: normal)

#1908: Direction signal of Spindle servo(0: reverse , non-0: normal)

#1923: Max scope of X-axis in negative direction (mm / inch)

#1924: Max scope of Y-axis in negative direction (mm / inch)

#1925: Max scope of Z-axis in negative direction (mm / inch)

#1926: Max scope of A-axis in negative direction (mm / inch)

#1927: Max scope of X-axis in positive direction (mm / inch)

#1928: Max scope of Y-axis in positive direction (mm / inch)

#1929: Max scope of Z-axis in positive direction (mm / inch)

#1930: Max scope of A-axis in positive direction (mm / inch)

#1950: G00 Speed of X-axis (mm/min)

#1950: G00 Speed of Y-axis (mm/min)

#1952: G00 Speed of Z-axis (mm/min)

#1953: G00 Speed of A-axis (mm/min or degree/min)

#1954: Default speed of G00(mm/min)

#1955: Simulate Speed(mm/min)

#1956: Max feeding speed(mm/min)

#1957: Acceleration of X-axis ((mm/min)/s)

#1958: Acceleration of Y-axis ((mm/min)/s)

#1959: Acceleration of Z-axis ((mm/min)/s)

#1960: Acceleration of A-axis((mm/min)/s)

#1960: Manual Speed of Spindle (rpm)

#1962: Manual Speed of Feeding axis (mm/min)

#1963: Speed of X-axis return to reference point in positive direction (mm/min)

#1964 : Speed of Y-axis return to reference point in positive direction (mm/min)

#1965: Speed of Z-axis return to reference point in positive direction (mm/min)

#1966: Speed of A-axis return to reference point in positive direction (mm/min)

#1967: Speed of X-axis return to reference point in negative direction (mm/min)

#1968: Speed of Y-axis return to reference point in negative direction (mm/min)

#1969: Speed of Z-axis return to reference point in negative direction (mm/min)

#1970: Speed of A-axis return to reference point in negative direction (mm/min)

#1970: Max speed of spindle (rpm)

#1972: Transmission Ratio of spindle in low grade(Motor_Speed/SP_Speed)

#1973: Transmission Ratio of spindle in high grade(Motor_Speed/SP_Speed)

#1974: Starting running speed in manual (mm/min)

#1975: Allow jump speed when continuous track interpolation(mm/min)

#1990: Number of processing workpieces

#1990: Times of cycle auto when using M20 code

#1992: Delay time when loose/tighten tool (s)

#1993: Tight tool auto after loose tool(0: no , non-0: yes)

#1994: Selection of Language

#1995: Direction of Q offset of G76 canned cycle

#1996: Direction of Q offset of G87 canned cycle

#1997: Width of track line in Diagram simulation (unit:pixel)

#1998: Mode of length compensation of tool

#1999: Type of spindle(0: servo spindle , Non-0: analog spindle)

#2000: Spindle is with Low/high grade (0: no , Non-0: yes)

#2000: Acceleration of spindle running normally

#2002: Turn on spindle when spindle change gear (0: no , Non-0: yes)

#2003: Time from stop to swing when spindle change gear (0.00s)

#2004: Time from swing to change gear when spindle change gear (0.00s)

#2005: Swing speed when spindle change gear (0.00 rpm)

#2006: Direction of starting swing when spindle change gear (0: CW, Non-0: CCW)

#2007: Time of swing CW when spindle change gear (0.00s)

#2008: Time of swing CCW when spindle change gear (0.00s)

#2009: Holding output signal that spindle change gear (0: no , Non-0: yes)

#2000: Hint user-defined alarm (0: no , Non-0: yes)

#2000: Hint alarm of feeding axis & spindle (0: no , Non-0: yes)

#2002: Hint alarm of coolant overload (0: no , Non-0: yes)

#2003: Hint alarm of coolant under-water (0: no , Non-0: yes)

#2004: Hint alarm of lubricate overload (0: no , Non-0: yes)

#2005: Hint alarm of lubricate under-oil (0: no , Non-0: yes)

#2006: Close function of 5 axis linkage interpolation (0: no , Non-0: yes)

#2007: Detect position feedback of spindle (0: no , Non-0: yes)

#2008: Use electric gear of spindle position feedback (0: no , Non-0: yes)

#2009: Control mode of spindle (G74/G84/G33 thread & spindle orientation)

#2020: Management way of tool(0: use M06, Non-0: use T code directly)

#2020: Way of selecting tool(0: single direction, Non-0: both direction random)

#2022: Way of feeding axes return to reference point

#2023: Max length of detecting zero position when homing

#2024: Max no. of tool case in fixed tool area

#2025: Reference point of Z-axis return when changing tool

#2026: Stopping degree when spindle orientation during tool change

#2027: Control mode when spindle homing

#2028: Power condition of each axis when booting

#2029: Max speed when Z-axis using handwheel (mm/min)

#2030: Max speed when X/Y/A-axis using handwheel (mm/min)

#2030: Detect SP_orientation & Point of changing point before change tool(Non-0: yes , 0: No)

#2046: stopping degree of spindle orientation when boring canned cycle

#2047: Homing Speed of spindle

#2048: Speed of spindle orientation

#2049: Allow Error Range of spindle orientation

#2050: Way of tool radius C compensation set up

#2050: Way of tool radius C compensation cancel

#2052: Position direction when SP-orientation adopt pulse interpolation control way

#2053: Delay time when spindle change direction suddenly (unit: 0.0s)

#2054: Acceleration of spindle servo ((mm/min)/s)

#2055: Active function of tool change

#2056: Transmission bits when communicate with RS232 serial port

#2057: Delay time between driver off of power & reboot (unit: 0.0s)

#2058: Interlock between motion of Z-axis and spindle take tool（0: No , Non-0:Yes)

#2059: Shifting distance after X-axis homing (unit:0.0mm)

#2060: Shifting distance after Y-axis homing (unit:0.0mm)

#2060: Shifting distance after Z-axis homing (unit:0.0mm)

#2062: Shifting distance after A-axis homing (unit:0.0mm)

#2070: Value of d quit tool in canned cycle G73 (mm)

#2070: Value of d quit tool in canned cycle G83 (mm)

#2000——#2099: Tool number of 0-99 tool case

#2032~#2045: Spare #2063~#2069: Spare

Format:#i=Expression

Could read the value, for example: #200=#1003; To read the X axis offset value of the third tool into macro variables #200.

Could modify the value, for example: #1003=23000; To modify the X axis offset value of the third tool to 23000 micron.

#1003=#1003+50; To increase the X axis offset value of the third tool 50 micron.

Macro internal register

3.3.4 Modal information (read only)

#4001 Modal information G00,G01,G02,G03,G33

#4002 Modal information G17,G18,G19

#4003 Modal information G90,G91

#4005 Modal information G94,G95

#4006 Modal information G20,G21

#4007 Modal information G40,G41,G42

#4008 Modal information G43,G44,G49

#4009 Modal information G73,G74,G76,G80-G89

#4010 Modal information G98,G99

#4014 Modal information G53,G54-G59

#4015 Modal information G60,G64

#4017 Modal information G15,G16

#4009 Modal information F

#4119 Modal information S

#4120 Modal information T

#4114 Sequence number

#4111 Modal information H

#4107 Modal information D

#7999 random number 1-9999

#8000 spindle encoder position, number of 0-4 frequency doubling encoder lines

#8004 spindle speed

3.3.5 Number of processed parts #3901 (read only)

3.3.6 Current status (read)

#5001--#5005 End point of current block's machine coordinate for XYZAB axis(mm)

#5021--#5025 Current position of XYZAB axis Machine coordinates values (mm)

#5041--#5045 Current position of XYZAB axis workpiece coordinates values (mm)

#5081 Current tool length compensation value

#5082 Current tool radius compensation value

#5080 Current tool number

#5091--#5095 corresponds to the current XYZAB axis workpiece coordinate system offset (mm)

#161--#165 Record the machine coordinates of the XYZAB axis after G31/G311 is executed (mm)

[#161-#5091]--[#165-#5095] corresponds to the current XYZAB axis workpiece coordinate (mm)

3.3.7 System parameter

#1001—#1099 X axis tool compensation, H1 length compensation value of tool T1--99 (um)

#1101—#1199 D1 radius compensation value of tool T1--99 (um)

#1201—#1299 Y/C axis tool compensation, H2 length compensation value of tool T1--99 (um)

#1301—#1399 D2 radius compensation value of tool T1--99 (um)

#1401—#1499 Z axis tool compensation, H3 length compensation value of tool T1--99 (um)

#1501—#1599 D3 radius compensation value of tool T1--99 (um)

#1601—#1699 A axis tool compensation, H4 length compensation value of tool T1--99 (um)

#1701—#1799 D4 radius compensation value of tool T1--99 (um)

#1800 input relay X00--07 (corresponding to D0--7)

#1801 Input relay X08--15 (corresponding to D0--7)

#1802 Input relay X16--23 (corresponding to D0--7)

#1803 Input relay X24--31 (corresponding to D0--7)

#1804 X, Y axis positive and negative limit X50--X57 (corresponding to D0--7)

#1805 Z, A-axis positive and negative limit X58--X65 (corresponding to D0--7)

#1806 B, C axis positive and negative limit X66--X73 (corresponding to D0--7)

#1807 Drive/spindle alarm X74--X81 (corresponding to D0--7)

#1808 Output relay Y00--Y15 (corresponding to D0--15)

#1809 Output relay Y16--Y31 (corresponding to D0--15)

#1810 Output relay Y32--Y47 (corresponding to D0--15)

#1811 Auxiliary relay M00--M15 (corresponding to D0--15)

#1812 Custom keyboard X101--X112 (corresponding to D0--11)

#1813 Servo enable Y50--Y57 (corresponding to D0--7)

#1814 Auxiliary relay M16--M31 (corresponding to D0--15)

#1815 Auxiliary relay M32--M47 (corresponding to D0--15)

#1816 Auxiliary relay M8--M63 (corresponding to D0--15)

#1817 Auxiliary relay M64--M79 (corresponding to D0--15)

#1818 Auxiliary relay M100--M115 (corresponding to D0--15)

#1819 Auxiliary relay M80--M95 (corresponding to D0--15)

#1820 Auxiliary relay M120--M135 (corresponding to D0--15)

#1850 X offset value of workpiece coordinate system G54 (unit: mm)

#1851 X offset value of workpiece coordinate system G55 (unit: mm)

#1852 X offset value of workpiece coordinate system G56 (unit: mm)

#1853 X offset value of workpiece coordinate system G57 (unit: mm)

#1854 X offset value of workpiece coordinate system G58 (unit: mm)

#1855 X offset value of workpiece coordinate system G59 (unit: mm

#1856 Y offset value of workpiece coordinate system G54 (unit: mm)

#1857 Y offset value of workpiece coordinate system G55 (unit: mm)

#1858 Y offset value of workpiece coordinate system G56 (unit: mm)

#1859 Y offset value of workpiece coordinate system G57 (unit: mm)

#1860 Y offset value of workpiece coordinate system G58 (unit: mm)

#1861 Y offset value of workpiece coordinate system G59 (unit: mm)

#1862 Z offset value of workpiece coordinate system G54 (unit: mm)

#1863 Z offset value of workpiece coordinate system G55 (unit: mm)

#1864 Z offset value of workpiece coordinate system G56 (unit: mm)

#1865 Z offset value of workpiece coordinate system G57 (unit: mm)

#1866 Z offset value of workpiece coordinate system G58 (unit: mm)

#1867 Z offset value of workpiece coordinate system G59 (unit: mm)

#1868 Workpiece coordinate system G54 A offset value (unit: mm)

#1869 Workpiece coordinate system G55 A offset value (unit: mm)

#1870 Workpiece coordinate system G56 A offset value (unit: mm)

#1871 Workpiece coordinate system G57 A offset value (unit: mm)

#1872 Workpiece coordinate system G58 A offset value (unit: mm)

#1873 Workpiece coordinate system G59 A offset value (unit: mm)

#1874 X value of the first reference point (unit: mm)

#1875 X value of the second reference point (unit: mm)

#1876 X value of the third reference point (unit: mm)

#1877 X value of the fourth reference point (unit: mm)

#1878 Y value of the first reference point (unit: mm)

#1879 Y value of the second reference point (unit: mm)

#1880 Y value of the third reference point (unit: mm)

#1881 Y value of the fourth reference point (unit: mm)

#1882 Z value of the first reference point (unit: mm)

#1883 Z value of the second reference point (unit: mm)

#1884 Z value of the third reference point (unit: mm)

#1885 Z value of the fourth reference point (unit: mm)

#1886 A value of the first reference point (unit: mm)

#1887 The A value of the second reference point (unit: mm)

#1888 The A value of the third reference point (unit: mm)

#1889 The A value of the fourth reference point (unit: mm)

#1890 Metric and inch setting of machine tool system

#1891 Fourth axis name

#1892 Fourth axis function setting

#1893 Rotation method when the fourth axis is a rotating axis

#1894 Relative coordinate display when the fourth axis is a rotating axis

#1895 Workpiece coordinate display when the fourth axis is a rotating axis

#1896 X-axis backlash compensation (micron)

#1897 Y-axis backlash compensation (micron)

#1898 Z axis backlash compensation (micron)

#1899 4th axis backlash compensation (micron)

#1900 Whether the feed axis uses electronic gears (0 means used/non-zero means not used)

#1901 X-axis electronic gear numerator (1-999999999)

#1902 The denominator of the X-axis electronic gear (1-999999999)

#1903 Y-axis electronic gear numerator (1-999999999)

#1904 Denominator of Y-axis electronic gear (1-999999999)

#1905 Z-axis electronic gear numerator (1-999999999)

#1906 The denominator of the Z-axis electronic gear (1-999999999)

#1907 The numerator of the fourth axis electronic gear (1-999999999)

#1908 The denominator of the fourth axis electronic gear (1-999999999)

#1909 Whether the spindle uses electronic gears (0 table use/non-zero table not use)

#1910 The numerator of the spindle low-end electronic gear (1-999999999)

#1911 The denominator of the low-end electronic gear of the 0main shaft (1-999999999)

#1912 The numerator of the high-end electronic gear of the main shaft (1-999999999)

#1913 The denominator of the high-end electronic gear of the main shaft (1-999999999)

#1914 X axis direction signal (0 means reverse/non-zero means normal)

#1915 Y-axis direction signal (0 means reverse/non-zero means normal)

#1916 Z axis direction signal (0 means reverse/non-zero means normal)

#1917 Fourth axis direction signal (0 means reverse/non-zero means normal)

#1918 Spindle pulse control direction signal (0 means reverse/non-zero means normal)

#1923 X-axis negative maximum stroke (mm/inch)

#1924 Y-axis negative maximum stroke (mm/inch)

#1925 Z axis negative maximum stroke (mm/inch)

#1926 The maximum travel of the fourth axis in the negative direction (mm/inch/deg)

#1927 X-axis positive maximum stroke (mm/inch)

#1928 Y-axis forward maximum stroke (mm/inch)

#1929 Z-axis forward maximum stroke (mm/inch)

#1930 The maximum stroke of the fourth axis in the positive direction (mm/inch/deg)

#1950 X axis G00 speed (mm/min)

#1951 Y axis G00 speed (mm/min)

#1952 Z axis G00 speed (mm/min)

#1953 Fourth axis G00 speed (mm/min or deg/min)

#1954 G01 Default speed (mm/min)

#1955 Simulation speed (mm/min)

#1956 Maximum feed rate (mm/min)

#1957 X-axis speed acceleration ((mm/min)/s)

#1958 Y axis speed acceleration ((mm/min)/s)

#1959 Z axis speed acceleration ((mm/min)/s)

#1960 4th axis speed acceleration ((mm/min)/s)

#1961 Spindle manual speed (rpm)

#1962 Manual speed of feed axis (mm/min)

#1963 X axis return to reference point positive speed (mm/min)

#1964 Y-axis reference point return positive speed (mm/min)

#1965 Z-axis reference point return positive speed (mm/min)

#1966 The forward speed of the fourth axis returning to the reference point (mm/min or deg/min)

#1967 X axis return to reference point reverse speed (mm/min)

#1968 Y-axis reference point return reverse speed (mm/min)

#1969 Z axis return to reference point reverse speed (mm/min)

#1970 Reverse speed of the fourth axis returning to the reference point (mm/min or deg/min)

#1971 Spindle motor maximum speed (rpm)

#1972 Spindle low gear ratio (motor speed/spindle speed)

#1973 Spindle high gear ratio (motor speed/spindle speed)

#1974 Initial speed during manual operation (mm/min)

#1975 Allowable speed jump variable for continuous trajectory interpolation (mm/min)

#1990 Setting of the number of parts to be processed

#1991 Program automatic running times setting when using M20 command

#1992 Delay time after tool loosening and tool tightening (seconds)

#1993 Whether to tighten the knife automatically after loosening the knife (non-zero means yes, 0 means no)

#1994 Language selection

#1995 Canned cycle G76 command offset Q direction

#1996 Canned cycle G87 command offset Q direction

#1997 Trace line width of graphics simulation (unit: pixel)

#1998 Tool length compensation method

#1999 Spindle type (0-meter servo spindle/non-zero-meter analog spindle)

#2000 Whether the spindle has high and low gears (0 means no/non-zero means yes)

#2001 The acceleration of the normal operation of the spindle ()

#2002 Whether the spindle is started when the spindle is shifted (0 means no/non-zero means yes)

#2003 The time from stop to swing when the spindle is shifted (0.01 second)

#2004 Spindle shift time from swing to shift (0.01 second)

#2005 Motor swing speed when the spindle shifts (0.01rpm)

#2006 The initial swing direction of the spindle when shifting gears (0 means positive/non-zero means reverse)

#2007 Spindle swing time when shifting gears (0.01 second)

#2008 Reverse swing time when the spindle is shifted (0.01 second)

#2009 Whether the spindle gear shift output signal is maintained (0 means no/non-zero means yes)

#2010 Whether to display custom alarms (0 means no/non-zero means yes)

#2011 Whether to display the feed axis and spindle alarm (0 means no/non-zero means yes)

#2012 Whether to display the cooling overload alarm (0 means no/non-zero means yes)

#2013 Whether to display the coolant shortage alarm (0 means no/non-zero means yes)

#2014 Whether to display lubrication overload alarm (0 means no/non-zero means yes)

#2015 Whether to display the lubrication shortage alarm (0 means no/non-zero means yes)

#2016 Whether to close the five-axis linkage interpolation function (0 means no/non-zero means yes)

#2017 Whether to detect spindle position feedback (0 means no/non-zero means yes)

#2018 Whether the spindle position feedback uses electronic gears (0 means no/non-zero means yes)

#2019 Spindle control mode (G74/G84/G33 thread processing and spindle positioning)

#2020 Tool management mode (0 table uses M06/non-zero table T command to change the tool directly)

#2021 Knife selection method (0 form to select a knife / non-zero form to select a knife randomly in both directions)

#2022 Feed axis reference point return method.

#2023 The maximum length of the zero signal to be detected when returning to zero

#2024 The largest tool pocket number in the fixed tool position area

#2025 Which reference point the Z axis returns to when changing the tool

#2026 Spindle orientation stop angle during tool change

#2027 Spindle control method when returning to zero

#2028 The power status of each axis when turning on

#2029 Z axis maximum speed when using MPG (mm/min)

#2030 X, the maximum speed of Y and the fourth axis when using MPG (mm/min)

#2031 Whether to detect the spindle orientation and Z-axis tool change point before tool change (non-zero table detection)

#2032 Reserve

#2033 Reserve

#2034 Reserve

#2035 Reserve

#2036 Reserve

#2037 Reserve

#2038 Reserve

#2039 Reserve

#2040 Reserve

#2041 Reserve

#2042 Reserve

#2043 Reserve

#2044 Reserve

#2045 Reserve

#2046 Spindle orientation stop angle of boring canned cycle

#2047 Spindle zero return speed

#2048 Spindle positioning speed

#2049 Spindle positioning error range

#2050 How to establish tool radius C compensation

#2051 How to cancel tool radius C compensation

#2052 The positioning direction when the spindle positioning adopts the pulse interpolation control method

#2053 The delay time when the main shaft turns to a sudden change (unit: 0.1 seconds)

#2054 Speed acceleration of spindle pulse control mode ((mm/min)/s)

#2055 Whether to start the tool selection and tool change function

#2056 Rs232 Serial communication rate

#2057 The delay time for the drive to be re-powered after power off (unit: 0.1 seconds)

#2058Z axis movement and master hand grasping tool interlocking (non-zero interlocking / zero-means interlocking)

#2059X axis offset distance after returning to the reference point (unit: 0.1mm)

#2060 Y-axis offset distance after returning to the reference point (unit: 0.1mm)

#2061Z axis offset distance after returning to the reference point (unit: 0.1mm)

#2062 The offset distance of the fourth axis after returning to the reference point (unit: 0.1mm)

#2063 Reserve

#2064 Reserve

#2065 Reserve

#2066 Reserve

#2067 Reserve

#2068 Reserve

#2069 Reserve

#2070 Canned cycle G73 command retraction amount d value (mm)

#2071 Canned cycle G83 command retraction value d (mm)

#2100-#2199 Tool number corresponding to tool set 0-99

#6000 Spindle first gear speed

#6001 Spindle second gear speed

#6002 Spindle third gear speed

#6003 Spindle fourth gear speed

#6007 Whether the spindle and the chuck interlocked

#7999 random number 1-999999

#8000 Spindle encoder position, 0-4 multiplier encoder lines

#8004 Spindle current speed

#8011-#8018 corresponds to the command position pulse of 8 feed axes

#8021-#8028 corresponds to the feedback position pulse of 8 feed axes

The following are the bus system and robot system coordinate system variables:

(1) When the unit is um:

#1850--#1855 G54--G59 X

#1856---#1861 G54-G59 Y

#1862--#1867 G54--G59 Z

#1868--#1873 G54--G59 A

#2090---#2095 G54--G59B

#2900---#2905 G54--G59C

#2910---#2915 G54-G59 XS

#2920--#2925 G54--G59 YS

#2501---#2548 G54.1-G54.48 X

#2551---#2598 G54.1--G54.48 Y

#2601---#2648 G54.1-G54.48 Z

#2651--#2698 G54.1--G54.48 A

#2701---#2748 G54.1--G54.48 B

#2751---#2798 G54.1--G54.48 C

#2801---#2848 G54.1--G54.48 XS

#2851---#2898 G54.1--G54.48 YS

#1874-#1877 1st-4th reference point X

#1878--#1881 1st-4th reference point Y

#1882---#1885 1st-4th reference point Z

#1886---#1889 1st-4th reference point A

#2096---#2099 1st-4th reference point B

#2906---#2909 1st-4th reference point C

#2916---#2919 1st-4th reference point XS

#2926---#2929 1st-4th reference point YS

(2) When the unit is mm:

#5221---#5228 G54 X-Y-Z-A-B-C-XS-YS

#5241---#5248 G55 X-Y-Z-A-B-C-XS-YS

#5261---#5268 G56 X-Y-Z-A-B-C-XS-YS

#5281---#5288 G57 X-Y-Z-A-B-C-XS-YS

#5301---#5308 G58 X-Y-Z-A-B-C-XS-YS

#5321---#5328 G59 X-Y-Z-A-B-C-XS-YS

Appendix I: Example of User-Defined M Macro Code

(1) Automatically Tool Setting

Macro variable of M880

X25: Input point , for detecting position of tool setting .

#380: Machine coordinate Value of X-axis Original Position Point for setting tool.

#380: Machine coordinate Value of Y-axis Original Position Point for setting tool.

#382: Machine coordinate Value of Z-axis Original Position & Return Point.

#383: Speed in negative direction of setting tool; (unit: mm/min).

#384: Speed in positive direction of setting tool; (unit: mm/min).

#385: Coordinate Value of Z-axis of surface of workpiece at current coordinate system after setting tool.

#386: Speed of moving to position point (unit: mm/min).

#387: Mode of Automatically Tool Setting (0: Fixed point for tool setting 0:Float point for tool setting) ; Fixed point for tool setting means that tool setting gauge is put at fixed position , XYZ axis will move to original position point before tool setting firstly at each time; Float point for tool setting means that Z-axis will move in negative direction to search for signal of tool setting signal.

#388: Min coordinate value in negative direction of Z-axis .

#389: Drop value of Z-axis,also height between surface of tool setting gauge & surface of workpiece .

#1925:Max scope of Z-axis in negative direction.

#1929:Max scope of Z-axis in positive direction.

#5023: Current machine coordinate value of Z-axis.

M Macro Program of " ProgramUser0 " as following:

```
STATUSINFO(Operating Tool Setting!);
IF (+#387) THEN;
MOVE(G90,F#386,Z#382);
MOVE(G90,F#386,X#380,Y#380);
ENDIF;
```

```
#50=#5023;
MOVE(G90,F#383,Z#388,W+25);//#1925
PAUS060;
IF (-X25) THEN
MESSAGEBOX(Error: Cannot detect signal of tool setting gauge!)
ERREXIT;
ENDIF
MOVE(G90,F#384,Z#1929,W-25);
IF (+X25) THEN
MESSAGEBOX(Error:Cannot off of signal of tool setting gauge!)
ERREXIT;
ENDIF
#50=#385+#389;
SETWK(Z#50);
MOVE(G90,F#386,Z#50);
MESSAGEBOX(Tool Setting Automatically Successfully!!)
RETURN;
```

(2) M882

```
STATUSINFO(Auto Set Drop Value of Z-axis of Tool Setting );
IF (+#387) THEN;
MOVE(G90,F#386,Z#382);
MOVE(G90,F#386,X#380,Y#380);
ENDIF;
MOVE(G90,F#383,Z#388,W+25);//#1925
PAUS060;
IF (-X25) THEN
MESSAGEBOX(Error:Cannot detect signal of tool setting gauge!)
ERREXIT;
ENDIF
```

```
MOVE(G90,F#384,Z#1929,W-25);
IF (+X25) THEN
MESSAGEBOX(Error:Cannot off of signal of tool setting gauge!)
ERREXIT;
ENDIF
#50=#5023;
MESSAGEBOX(Move Tool Tip to Surface of Workpiece & Run M883 code!)
RETURN;
```

(3) M883

```
#389=#50-#5023;
MESSAGEBOX(Drop of Z-axis Set Well for setting tool!)
RETURN;
```

(4) Steps of Automatically Tool Setting

① Set P380~P388 in Other Parameter

② Set Drop of Z-axis of tool setting automatically

a. Run M882 command in MDI , Prepare set drop

b. Move tool tip of Z-axis to surface of workpiece manually

c. Run M883 command in MDI, set drop of Z-axis automatically(P389)

③ Select workpiece coordinate system(G54~G59)

④ Automatically Tool Setting: Run M880 in MDI to set offset of Z-axis at current workpiece coordinate system.

Appendix II: Example of User-Defined G Macro Code

For example, defines the G052 function: the arc model porous drilling cycle. (must copy the macro program ProgramG052 into system).

Format:G052 Xx Yy Zz Rr Ii Aa Bb Hh Ff;

X: The X coordinate with absolute value or incremental value of center to specify.

Y: The Y coordinate with absolute value or incremental value of center to specify.

Z: Hole depth

R: Approaching fast to the point coordinate

F: Cutting feed speed

I: Radius

A: The angle of the first hole

B: Incremental angle specify(CW when negative)

Macro program ProgramG052 as follows:

```
#80=#0
#81=#1
#82=#2
#83=#3
#84=#4
#85=#5
#86=#6
#87=#7
#88=#8
#89=#9
#90=#10
#90=#10
#92=#12
#93=#13
#94=#14
#95=#15
```

```
#96=#16
#97=#17
#98=#18
#99=#19
#100=#20
#30=#4003
#30=#4004
G90
IF[#30 EQ 90] GOTO 0
G53
#98=#5000+#98
#99=#5002+#99
N0 WHILE[#86 GT 0] DO 0
#35=#98+#87*COS[#80]
#36=#99+#87*SIN[#80]
G80X#35Y#36Z#100R#92F#85
#80=#80+#80
#86=#86-0
END 0
G#30 G#30 G80
M99
```