

User Manual

Macro of CNC System

V2.0

Shenzhen Guanhong Automation CO.,LTD

Website: www.szghauto.com

Add:Room 503 Anxin Building, No 536 Shenhui Road, Liuyue community, Henggang Street ,Longgang District, Shenzhen City,Guangdong Province, ChinaProvince, China

Post code: 518115

-Catalog-

Chapter 1	MACRO.....	1
1.1	Non-Mode Macro Command G65.....	1
1.2	Mode Macro Command G66/G67.....	2
Chapter 2	M MACRO.....	3
2.1	User-defined M Macro Program(M881-M889).....	3
2.2	Instructions of M Macro Program(M881-M889).....	3
2.2.1	Output Instruction: OUT +/- Y/M.....	3
2.2.2	Wait Instruction: WAT +/- X/Y/M.....	3
2.2.3	Delay Instruction: PAUS.....	4
2.2.4	Assignment Instruction: =.....	4
2.2.5	Information Prompt Dialog.....	4
2.2.6	Conditional Statement.....	4
2.2.7	Move Coordinate Axes.....	5
2.2.8	Set & Save Current Workpiece Coordinate.....	5
2.2.9	Set & Save Current Machine Coordinate(G53).....	6
2.2.10	Error & Exit.....	6
2.2.11	Return.....	6
Chapter 3	G MACRO.....	7
3.1	User-defined G Macro Program.....	7
3.2	Macro program instruction.....	7
3.2.1	Input Instruction: WAT.....	7
3.2.2	Output Instruction: OUT.....	7
3.2.3	Assignment Instruction: =.....	7
3.2.4	Unconditional Jump: GOTO n.....	7
3.2.5	Conditional Jump.....	8
3.2.6	Loop Command.....	9
3.3	Operators' meaning.....	10
3.4	Arithmetic & Logic Operation.....	10
Chapter 4	Variable.....	11
4.1	Local Variable.....	11
4.2	Global Variable.....	11
4.3	System Variable.....	11
4.3.1	System Parameter Variable.....	11
4.3.2	I/O variables.....	11
Chapter 5	Dialog Box of Macro Variable.....	19
5.1	Steps of Setting Dialog Box.....	19
5.2	Configure of Dialog Box.....	20
Appendix I:	Example of User-Defined M Macro Code.....	23
Appendix II:	Example of User-Defined G Macro Code.....	26

Chapter 1 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

Special Note: Address G , L , N , Q , P cannot used as arguments.

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

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		

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(M881-M889)

2.2.1 Output Instruction: OUT +/- Y/M

Explanation: Set output Y, M is valid or invalid

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

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.

2.2.2 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.

Attention: "+" to means wait input is effective; "-" means wait input is invalid;

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

2.2.3 Delay Instruction: PAUS

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

Eg.: PAUS100 : delay 100 ms .

PAUS1000 : delay 1 second

2.2.4 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

2.2.5 Information Prompt Dialog

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

2.2.6 Conditional Statement

Conditional statement: when condition is met, excutive codes after THEN, otherwise jump conditonal codes.

Format: IF(**) THEN

ENDIF

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

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

-M** : When auxilary 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

E.g.2: When macro variable #313 is 1(non-0) , pop up dialog and exit.

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

2.2.7 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.

2.2.8 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 coordiante system ,set value of X-axis to #238 , set value of Z-axis to #237.

2.2.9 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 ;

2.2.10 Error & Exit

Format: ERREXIT

When macro program have errors, it will exit macro program

2.2.11 Return

Format: RETURN

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

Chapter 3 G MACRO

3.1 User-defined G Macro Program

Program Names of G120-G160 , user-defined G codes, are corresponding to ProgramG120 - ProgramG160 .

G Macro program(ProgramG120 - ProgramG160) must be edited on computer, and then copy to U-disk , and restore into cnc system.

3.2 Macro program instruction

3.2.1 Input Instruction: WAT

Waiting for the input port X valid or invalid instruction

Format: WAT+ (-) X

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

"-" means the input is invalid;

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

3.2.2 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;

3.2.3 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

3.2.4 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

3.2.5 Conditional Jump

1) IF (Conditional express) GOTO n

If condition is met, execute GOTO n ,jump to N** program line; if the condition isnot 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 opertional 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

3.2.6 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 isnot 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

```

3.3 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

3.4 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 Cos Acos Tan Atan	#i = SIN(#j) ; #i = ASIN(#j); #i = COS(#j) ; #i = ACOS(#j); #i = TAN(#j); #i = ATAN(#j);	90.5 degrees means 90 degrees & 30 points
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

Chapter 4 Variable

4.1 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.

4.2 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.

4.3 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.

4.3.1 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)

4.3.2 I/O variables

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

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

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

#1802: X16-X23 (D0-D7) ; input resistor
#1803: X24-X31 (D0-D7) ; 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-X112(D0-D11) ; input of operational panel
#1813: Y50-Y57(D0-D7) ; servo enable
#1814: M16-M31(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-D15) ; auxiliary relay
#1850: X-axis Offset value of G54 workpeice coordinate system(unit:mm)
#1851: X-axis Offset value of G55 workpeice coordinate system(unit:mm)
#1852: X-axis Offset value of G56 workpeice coordinate system(unit:mm)
#1853: X-axis Offset value of G57 workpeice coordinate system(unit:mm)
#1854: X-axis Offset value of G58 workpeice coordinate system(unit:mm)
#1855: X-axis Offset value of G59 workpeice coordinate system(unit:mm)
#1856: Y-axis Offset value of G54 workpeice coordinate system(unit:mm)
#1857: Y-axis Offset value of G55 workpeice coordinate system(unit:mm)
#1858: Y-axis Offset value of G56 workpeice coordinate system(unit:mm)
#1859: Y-axis Offset value of G57 workpeice coordinate system(unit:mm)
#1860: Y-axis Offset value of G58 workpeice coordinate system(unit:mm)

#1861: Y-axis Offset value of G59 workpeice coordinate system(unit:mm)
#1862: Z-axis Offset value of G54 workpeice coordinate system(unit:mm)
#1863: Z-axis Offset value of G55 workpeice coordinate system(unit:mm)
#1864: Z-axis Offset value of G56 workpeice coordinate system(unit:mm)
#1865: Z-axis Offset value of G57 workpeice coordinate system(unit:mm)
#1866: Z-axis Offset value of G58 workpeice coordinate system(unit:mm)
#1867: Z-axis Offset value of G59 workpeice coordinate system(unit:mm)
#1868: A-axis Offset value of G54 workpeice coordinate system(unit:mm)
#1869: A-axis Offset value of G55 workpeice coordinate system(unit:mm)
#1870: A-axis Offset value of G56 workpeice coordinate system(unit:mm)
#1871: A-axis Offset value of G57 workpeice coordinate system(unit:mm)
#1872: A-axis Offset value of G58 workpeice coordinate system(unit:mm)
#1873: A-axis Offset value of G59 workpeice coordinate system(unit:mm)
#1874: X-axis Coordinate Value of 1st refenerce point (unit:mm)
#1875: X-axis Coordinate Value of 2nd refenerce point (unit:mm)
#1876: X-axis Coordinate Value of 3rd refenerce point (unit:mm)
#1877: X-axis Coordinate Value of 4th refenerce point (unit:mm)
#1878: Y-axis Coordinate Value of 1st refenerce point (unit:mm)
#1879: Y-axis Coordinate Value of 2nd refenerce point (unit:mm)
#1880: Y-axis Coordinate Value of 3rd refenerce point (unit:mm)
#1881: Y-axis Coordinate Value of 4th refenerce point (unit:mm)
#1882: Z-axis Coordinate Value of 1st refenerce point (unit:mm)
#1883: Z-axis Coordinate Value of 2nd refenerce point (unit:mm)
#1884: Z-axis Coordinate Value of 3rd refenerce point (unit:mm)
#1885: Z-axis Coordinate Value of 4th refenerce point (unit:mm)
#1886: A-axis Coordinate Value of 1st refenerce point (unit:mm)
#1887: A-axis Coordinate Value of 2nd refenerce point (unit:mm)
#1888: A-axis Coordinate Value of 3rd refenerce point (unit:mm)
#1889: A-axis Coordinate Value of 4th refenerce point (unit:mm)
#1890: Set of Metric/Inch in system

- #1891: 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 Compensatoin of X-axis (unit: um)
- #1897: Reverse Backlash Compensatoin of Y-axis (unit: um)
- #1898: Reverse Backlash Compensatoin of Z-axis (unit: um)
- #1899: Reverse Backlash Compensatoin of A(4th)-axis (unit: um)
- #1900: Feeding axis use electric gear(0: use , non-0: no)
- #1901: Numerator of X-axis's electron gear (1—999999999)
- #1902: Denominator of X-axis's electron gear (1—999999999)
- #1903: Numerator of Y-axis's electron gear (1—999999999)
- #1904: Denominator of Y-axis's electron gear (1—999999999)
- #1905: Numerator of Z-axis's electron gear (1—999999999)
- #1906: Denominator of Z-axis's electron gear (1—999999999)
- #1907: Numerator of A-axis's electron gear (1—999999999)
- #1908: Denominator of A-axis's electron gear (1—999999999)
- #1909: Spindle use electric gear (0: use , non-0: no)
- #1910: Numerator of electron gear ratio of spindle in Low grade (1—999999999)
- #1911: Denominator of electron gear ratio of spindle in Low grade (1—999999999)
- #1912: Numerator of electron gear ratio of spindle in Low grade (1—999999999)
- #1913: Denominator of electron gear ratio of spindle in High grade (1—999999999)
- #1914: Direction signal of X-axis(0: reverse , non-0: normal)
- #1915: Direction signal of Y-axis(0: reverse , non-0: normal)
- #1916: Direction signal of Z-axis(0: reverse , non-0: normal)
- #1917: Direction signal of A-axis(0: reverse , non-0: normal)
- #1918: 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)
- #1951: 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 G01(mm/min)
- #195: 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)
- #1961: Manual Speed of Spindle (rpm)
- #1962: Manual Speed of Feedinga 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)
- #1971: 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
- #1991: 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 Lanuage
- #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)
- #2001: 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 (0.01s)
- #2004: Time from swing to change gear when spindle change gear (0.01s)
- #2005: Swing speed when spindle change gear (0.01 rpm)
- #2006: Direction of starting swing when spindle change gear (0: CW, Non-0: CCW)
- #2007: Time of swing CW when spindle change gear (0.01s)
- #2008: Time of swing CCW when spindle change gear (0.01s)
- #2009: Holding output singal that spindle change gear (0: no , Non-0: yes)
- #2010: Hint user-defined alarm (0: no , Non-0: yes)
- #2011: Hint alarm of feeding axis & spindle (0: no , Non-0: yes)
- #2012: Hint alarm of coolant overload (0: no , Non-0: yes)
- #2013: Hint alarm of coolant under-water (0: no , Non-0: yes)
- #2014: Hint alarm of lubricate overload (0: no , Non-0: yes)
- #2015: Hint alarm of lubricate under-oil (0: no , Non-0: yes)
- #2016: Close function of 5 axis linkage interpolation (0: no , Non-0: yes)
- #2017: Detect position feedback of spindle (0: no , Non-0: yes)

- #2018: Use electric gear of spindle position feedback (0: no , Non-0: yes)
- #2019: Control mode of spindle (G74/G84/G33 thread & spindle orientation)
- #2020: Managment way of tool(0: use M06, Non-0: use T code directly)
- #2021: 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)
- #2031: 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
- #2051: 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: 0.1s)
- #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: 0.1s)
- #2058: Interlock between motion of Z-axis and spindle take tool (0: No , Non-0:Yes)
- #2059: Shifting distance after X-axis homing (unit:0.1mm)
- #2060: Shifting distance after Y-axis homing (unit:0.1mm)

#2061: Shifting distance after Z-axis homing (unit:0.1mm)

#2062: Shifting distance after A-axis homing (unit:0.1mm)

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

#2071: Value of d quit tool in canned cycle G83 (mm)

#2100——#2199: 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.

Chapter 5 Dialog Box of Macro Variable

In order to support better satisfaction and let users can operate SZGH CNC system more easily, SZGH CNC system can support dialog box of macro variable that can check and set macro variable.

Files of “interfacecn.cfg” & “interfaceeg.cfg” is used for set dialog box of macro variable.

“interfacecn.cfg” is set for displayed on Chinese screen

“interfaceeg.cfg” is set for displayed on English screen.


Note: 1. Files of “interfacecn.cfg” & “interfaceeg.cfg” must be edited by Windows Notebook or CNC system.

2. When first word is “/” is someone line, which means that the line is for explanation

3. All dialog box of macro variable are edited on files of “interfacecn.cfg” & “interfaceeg.cfg”.

4.A group of “BeginMainFace”& “EndMainFace” means one interface.

5.1 Steps of Setting Dialog Box

- ① Finish & Design related files & pictures
- ② Restore related files(interfacecn.cfg , interfaceeg.cfg & configuration of parameters) & pictures into cnc system
- ③ Set P900 in Other parameter to 4 , set to user-defined processing parameter dialog box
- ④ Reboot CNC system
- ⑤ Press “Enter”key in main screen to enter Dialog Box of Macro variable
- ⑥ Press “PgDn” & “PgUp” to select related interface of dialog box
- ⑦ Press “

19

5.2 Configure of Dialog Box

Start from “ BeginMainFace ” , End of “ EndMainFace ” .

A pair of “BeginMainFace”& “EndMainFace” means one dialog box of macro variable.

Format:

```
BeginMainFace
T1,L1,T1,W1,H1,M1,I0,
...
Tn,Ln,Tn,Wn,Hn,Mn,In,
EndMainFace
```

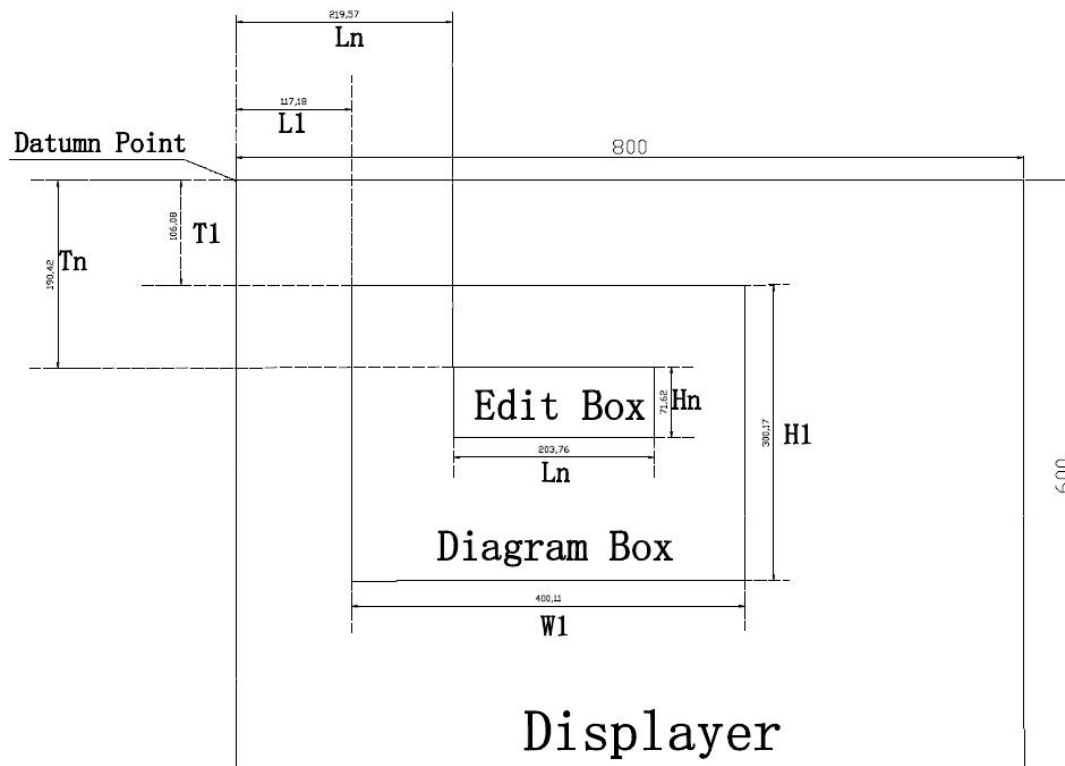
(1) BeginMainFace: Start configuring of dialog box

(2) T1~Tn: 1st data, Controls’ Type , Range: 0 ~ 4 ;

0: Dialog Box ; 1: User-defined Static Text ; 2: User-defined Button ; 3:

User-defined Edit box ; 4: Picture

(3) Dimension of Box: Left/Top/Width/Height (Schematic Diagram)



Size & Position of Dialog: datumn point is based on point of upper left corner ,

Unit: Pixels. (Displayer is 800*600 pixels)

Left: L1~Ln: 2nd data, start from datum point, left to left side of dialog box ;

Top: T1~Tn: 3rd data, start from datum point, down to top side of dialog box;

Width: W1~Wn: 4th data, width of dialog box

Height: T1~Tn: 5th data, height of dialog box

(4) If 1st data is “0”, which means configuration of dialog box, and we could set position & size of dialog box of macro variable; 5th data can be omitted;

If 1st data is “1”, which means configuration of static text, normally used as prompt informations, 5th data display the prompt information;

If 1st data is “2”, which means configuration of pressing button, 5th data is for displaying prompt messages of button;

If 1st data is “3”, which means configuration of edit box , also used for assignment of macro variable, 5th data is for specify macro variable; 6th data is used for specify type of this macro variable, “I” means integer type, no specify means floating-point type;

If 1st data is “4”, which means configuration of picture, 5th data is the name of picture ,

Name of Picture: fillpict1, fillpict2, fillpict3 ...

Format of Picture: 16-color (4 bits) or 256-color(8bits) or 16bits or 24 bits BMP type formatted image

Note: Please try to use 4bits or 6bits pictures to save occupied memory of picture.

(5) EndMainFace: End of configuration of current interface of dialog box.

Example:

BeginMainFace //start configure dialog box

0,5,30,450,390, //4 datas stands for Left/Top/Width/Height of dialog box

4,3,3,280,190,fillpict1.bmp // fillpict1.bmp,name of displaying diagram

1,282,10,60,25,X: //1st data: control type, following 4 datas stand for Left/Top/Width/Height , last data: number of related macro variable(only for edit box) or hint message

3,330,10,85,25,345,I // #345 is macro variable, type is integer

1,282,40,60,25,Z:

3,330,40,85,25,288, // #288 is macro variable, type is float-point type

1,282,70,60,25,R:

3,330,70,85,25,500,I

1,282,100,60,25,P:

3,330,100,85,25,700,

1,282,130,60,25,S:

3,330,130,85,25,456,

1,200,195,100,25,Pitch K(E):

3,300,195,90,25,145,

1,200,223,100,25,Time L:

3,300,223,90,25,225,

2,30,253,100,26,Enter

2,240,253,100,26,ESC

EndMainFace // End of configuring

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.

#381: 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 Automatially Tool Setting (1: 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: Curren 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#381);

ENDIF;

```
#50=#5023;
MOVE(G90,F#383,Z#388,W+25);//#1925
PAUS160;
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
#51=#385+#389;
SETWK(Z#51);
MOVE(G90,F#386,Z#50);
MESSAGEBOX(Tool Setting Automatically Succesfully!!)
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#381);
ENDIF;
MOVE(G90,F#383,Z#388,W+25);//#1925
PAUS160;
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 G152 function: the arc model porous drilling cycle.
(must copy the macro program ProgramG152 into system).

Format:G152 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 ProgramG152 as follows:

#80=#0

#81=#1

#82=#2

#83=#3

#84=#4

#85=#5

#86=#6

#87=#7

#88=#8

#89=#9

#90=#10

#91=#11

#92=#12

#93=#13

#94=#14

#95=#15

```
#96=#16
#97=#17
#98=#18
#99=#19
#100=#20
#30=#4003
#31=#4014
G90
IF[#30 EQ 90] GOTO 1
G53
#98=#5001+#98
#99=#5002+#99
N1 WHILE[#86 GT 0] DO 1
#35=#98+#87*COS[#80]
#36=#99+#87*SIN[#80]
G81X#35Y#36Z#100R#92F#85
#80=#80+#81
#86=#86-1
END 1
G#30 G#31 G80
M99
```