New 990TDB Operation Manual

Edited and extracted from original manuals - Steve Blackmore September 2016
If you find any errors or have further information please contact me via email steve@pilotltd.net

Chapter 1 Preface

This device is a universal CNC Lathe control system developed by our company. The control uses the latest industrial high-speed ARM processor, large-scale field programmable (FPGA) technology, and multilayer PCBs. It is constructed using the latest highly integrated chip and surface mount components to provide a very reliable and adaptable control system in a compact and reasonably priced package.

The controller provides real-time control at high speed (Rapid Speeds up to 30 M/min, and interpolation speeds of 15 M/min), with high precision. The 800x600 dot matrix TFT LCD and adaptive brightness LED backlight maintains display uniformity, and provides a long service life by adjusting the screen to suit environmental temperature changes. With a full screen English menu display, operation is simple and convenient.

- This system is based on a lathe as representative of a two, three, or four axis closed-loop fully digital universal control system. It combines powerful functions and a broad instruction set, with programming code in accordance with ISO (International Code) standards. It can be easily adapted for direct control of AC servos by combining it with suitable single or dual channel AC Servo drivers.
- This manual details the programming and use for a lathe control system.

Note for “Caution”:

1/ “**Caution**” Reminds the operator to be cautious in the relative operation，otherwise the operation will fail or some action cannot be completed.

2/ “**Special Caution**” Warns the operator to use Special Caution in the relative operation otherwise it may damage the machine or give rise to an accident.

Special hint：

This system has a dedicated function to back-up the machine parameters. After debugging the machine, all parameters of the machine, system, and PLC documents can easily be backed up to a computer. This is convenient not only for major debugging, but also for machine recovery to a previous state after system changes.

Note ：

Before using this system for the first time, to improve safety and efficiency, please read all of the details in each chapter carefully.

Chapter 2 Programming

Programming refers to the process of using CNC language to describe a machining path and actions based on the part machining blueprint and technique requirements.

2.1 Basic concepts

Program Line: A complete single command line consisting of instruction segments and data segments.

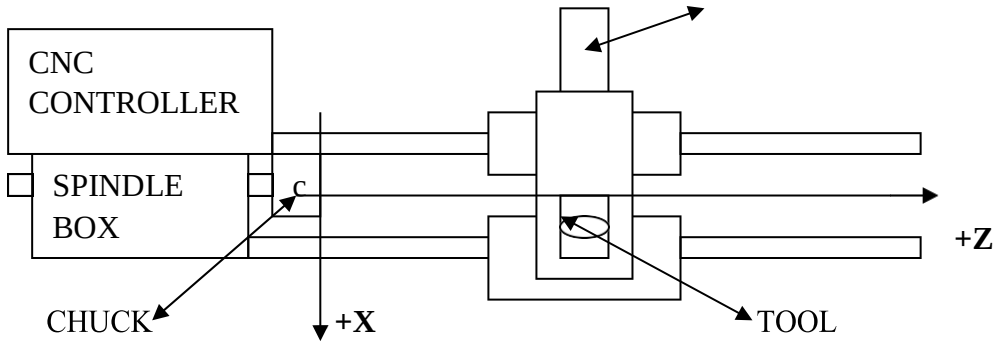
Program: A congregation of sequential program lines in a logical order to complete the machining of a workpiece.

Machine Coordinate System: Refer fig. "Lathe top view:" below.

Machine coordinates stipulate the machine coordinate and direction, Z axis parallels the spindle, with the direction away from spindle nose as Z+ (positive). The X axis is perpendicular (90 degrees) to the spindle centre line, with movement away from the chuck normally as X+ (positive).

Lathe top view:

SERVO MOTOR



Absolute programming: This is the confirmed coordinate data programming mode based on an established absolute coordinate system of X & Z.

Relative programming (Incremental programming): The distance and direction of an operation end point, relative to the current starting point. Uses U & W.

Mixed programming: Is the combination of absolute coordinates and relative coordinates in the one command line of a program.

Diameter programming: All X axis coordinates are presented as diameters.

Starting point: The initial position of the active tool when a program starts.

Name of program: The name of the machining program.

Modal instruction: An instruction which can remain as the active function in a program. It remains active until cancelled or overridden by another modal command. In the same program, but on separate lines, there may be several modal commands, such as M03 (spindle clockwise), M04 (spindle counter clockwise), M05 (spindle stop). These are all Modes used to control spindle. Modes of the same kind are categorized into modal groups. At any time there can only be one of each group active. The initial active mode command is called the mode origin. In the above modal group, M05 is the mode origin. This ensures that the spindle is stopped when the machine is first turned on.

Suspending mode (removing mode): This is a command that can turn a mode command into the mode origin or cancel the current mode. For example, M20 (program ending command) causes the end of an operation and returns the controller to the original status.

Non-modal command: This is a command which has no function to store, and is only active in that line of the program.

2.2 Program commands (Commands)

2.2.1 Functional meaning of address symbols, and data list.

Functions	Address symbol	Meaning	Data range
Program line No	N	No of program line	
Preparation function	G	Content and mode of designated command operation.	00-99
Auxiliary function	M	Auxiliary operation instruction	00-99
Tool function	T	Tool change command	Tool T0101-9999
Spindle function	S	The primary spindle speed	0-65000
Spindle function	SS	The secondary spindle speed	0-65000
Cutting speed	F	Interpolation Feed per minute, per rotation.(Dependent upon G98-99)	1-30000mm/min, 0.001-9.999mm/n
Coordinates character	X(U) Z(W)	The coordinate values of the X Z axes	±99999.9999mm
Lead of screw thread	F(I)	F for metric, I for imperial	0.1-1000mm, 1-99 threads/inch
Core coordinates	I, K	X & Z axes incremental coordinate values for centers of Arcs	±99999.9999mm
Arc radius	R	Arc radius value, Tool radius value	0.001-99999.999mm
Delay time	XUP	Period of programmed delay	0.001-65s
Program entrance	P	Entrance of calling program line	0000-99999
Repeat times	L	Repeats for cycle or subprogram calls. L can be used for number of multiple starts in screw threads.	1-65000 1-99
Program skip	/	If there is an "/" before N this line does not run	

2.2.2 Program command table

Group	Nature	Code	Function	Origin	Mode	Remarks
1	Main Control Functions	G00	Rapid Traverse positioning		√	
		G01	Linear interpolation		√	
		G02	Circular—clockwise		√	
		G03	Circular—counterclockwise		√	
		G32	Spiral interpolation			
		G332	G02 spiral			
		G333	G03 spiral			
		G31	Check & jump on No alarm			
		G311	Check & jump on alarm			
	Loop Commands	G70	Finish machining loop		√	
		G71	Cylindrical face thick loop		√	
		G72	End face thick loop		√	
		G73	Close loop			
		G74	End face deep hole loop		√	
		G75	Slot loop		√	
		G76	Complex screw thread loop			
		G90	Circular loop		√	
		G92	Screw thread loop		√	
		G93	Tapping cycle		√	
		G94	End face loop		√	
		G22	Loop end		√	
		G800	Cancel loop		√	
	Go To Start	G26	X & Z, Go to start X,			
		G261	Go to start			
		G262	Y, Go to start Z,			
		G263	Go to start A,			
		G264	Go to start			
	Go G25	G61	X & Z, Go to G25 X,			
		G611	Go to G25Y,			
		G612	Go to G25 Z,			
		G613	Go to G25 A,			
		G614	Go to G25			

	Save	G25	Save current coordinates		√	
	Go Home	G28 G281 G282 G283 G284 M800	X or Z, Home X, Go to Home Y/C, Go to Home Z, Go to Home A, Go to Home C, Go to encoder Zero		√	
	Set	G50	Setup co-or system		√	
		G52	Setup part co-or system		√	
		G184	Setup current Tool co-or			
		G185	Setup all Tool co-or			
2	Line	G96	Constant line speed cutting		√	
		G97	Cancel	√	√	
	Feed	G98	Set feed per minute	√	√	
		G99	Set feed per revolution		√	
	Mode	G15	Cancel	√	√	
		G16	Ball co-or program		√	
		G21	Use Metric coordinates	√	√	
		G20	Use Imperial coordinates		√	
3	Delay	G04	Programmed dwell			
4	Cut	G60 G64	Precise end point Continuous path	√	√ √	
5	Compensation	 G40 G41 G42	 Cancel cutter compensation Cutter to left of workpiece Cutter to right of workpiece	√	√ √ √	
6	Work Coordinates	G53 G54 G55 G56 G57 G58 G59	Machine co-or Work co-or 1 Work co-or 2 Work co-or 3 Work co-or 4 Work co-or 5 Work co-or 6	√	√ √ √ √ √ √ √	

7	Spindle	M03 M04 M05	Spindle clockwise run Spindle counterclockwise Spindle off	 √	√ √ √	
8	Coolant	M08 M09	Coolant on Coolant off	 √	√ √	
9	Chuck	M10 M11	Chuck clamp Chuck un-clamp	√	√ √	
10	Tailstock	M79 M78	Extend Retract	 √	√ √	M79
11	Lubrication	M32 M33	Lubrication on Lubrication off	 √	√ √	M32
12	Air	M59 M58	Air on Air off	 √	√ √	M59 output
13	User Defined Options	M61 M60	user-defined 1		√ √	M61 output
		M63 M62	user-defined 2		√ √	M63 output
		M65 M64	user-defined 3		√ √	M65 output
		M67 M66	user-defined 4		√ √	M67 output

		M69 M68	user-defined 5		√ √	M69 output
		M71 M70	user-defined 6		√ √	M71 output
		M73 M72	user-defined 7		√ √	M73 output
		M75 M74	user-defined 8		√ √	M75 output
14	User Defined Input	M12 M13	Check M12 valid Check M12 invalidate		√ √	M12 INPUT
		M14 M15	Check M14 valid Check M14 invalidate		√ √	
		M16 M17	Check M16 valid Check M16 invalidate		√ √	
		M18 M19	Check M18 valid Check M18 invalidate		√ √	
		M28 M29	Check M28 valid Check M28 invalidate		√ √	
		M22 M23	Check M22 valid Check M22 invalidate		√ √	
		M24 M25	Check M24 valid Check M24 invalidate		√ √	
15	Spindle Gear	M41 M42 M43 M44	Select Spindle Gear 1 Select Spindle Gear 2 Select Spindle Gear 3 Select Spindle Gear 4			
16	Jump	M97 M98 M99	Jump to Subroutine Call Subroutine Return from Subroutine			
17	Program	M00 M01 M02 M30 M20	Pause process If M22 active, pause process Program end M05, M09, and End process Loop, Go to Start			
18	Spindle	S SS	First Spindle Speed Second Spindle Speed		√ √	S=0-650 00 SS=0-65 000
19	Tool	Tab	A:xx B:xx		√	A、 b=00-12

2.3 Preparation functions

2.3.1 Programming conditions:

1, Multiple commands are permitted on the same line, but commands of the same Modal Group cannot share one line.

2, Commands and parameters can be arranged in any order within a program line. (Such as: G01 U10 W-30 can be written: U10 G01 W-30)

3, Commands must not be repeated within a program line.

4, Irrelevant parameters and operations must not be included in any line.

5, It is permissible to omit the "0" before some commands. For example, G01 G03 can be written as G1 G3.

6, The command of an optional point, line start, or that after a tool change commands must be programmed using absolute coordinates.

2.3.2 Commands:

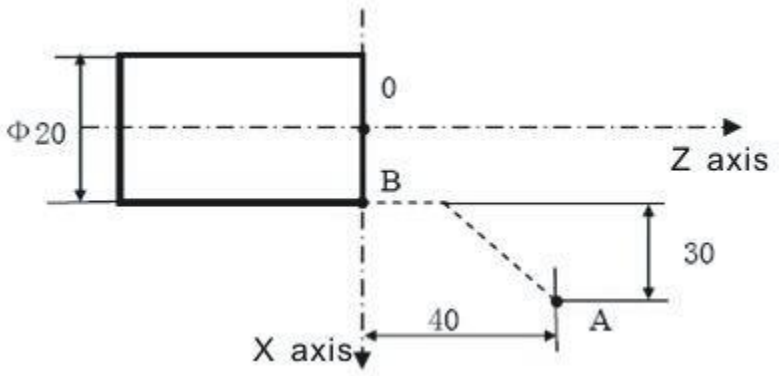
（1）Rapid Traverse (G00)

Move tool to desired new positon at high speed set in parameter (G00). In Absolute coordinates, program absolute X and Z end point；

In incremental mode, program relative (U, V & W) coordinates of end point.

Format：G00 X/U- Y/V- Z/W- A- (Mode，original)

Note: Y, Z, and A means a motion axis. The command and data indicate the distance and direction by either the absolute or incremental method。 G00 moves the tool in a straight line directly to the programmed point。 Rapid speed



is determined by the G00 parameter。

Example：from A to B.

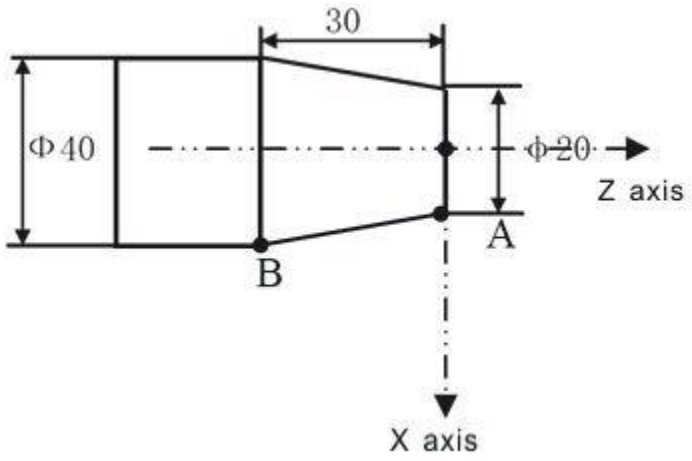
Absolute program: G00 X20 Z0； Relative program: G00 U－60 W－40；

（2）Linear Interpolation (G01)

Used for single axis motion or 2, 3, or 4 axis interpolation motion。

Format：G01 X/U- Y/V- Z/W- A- F- (Mode)

Note: X, Y, Z, or A means a motion axis. The data defines a motion distance and direction by an absolute or incremental method. Motion speed is determined by a commanded F word. The F instruction is modal.



Example：from A to B.

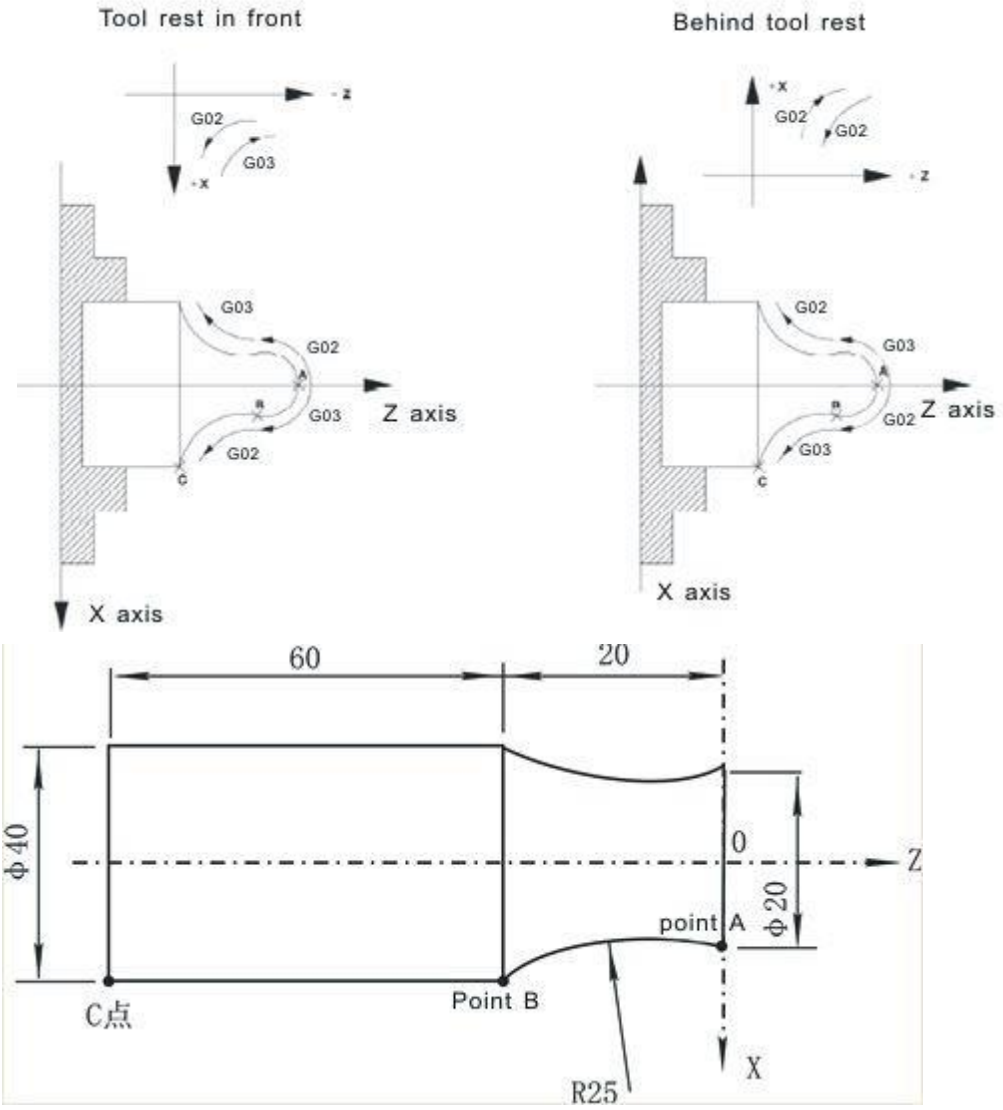
Absolute program: G01 X40 Z-30 F100 Relative program: G01 U20 W-30 F100

（3）Arc interpolation (G02/G03)

G02 is the command used for clockwise Circular interpolation, and G03 for counter-clockwise Circular interpolation. The X(U) and Z(W) coordinates are the exact end point of the arc, I is the incremental distance in the X axis from the starting point of the arc to the centre of the arc, and K is the incremental distance in the Z axis from the starting point of the arc to the centre of the arc. The sign (+/-) of the I and K components is critical to the command. It can be also programmed using R instead of I & K.

Format：

G02 X（U）__ Z（W）__ I__ K__ F__； G03 X（U）__ Z（W）__ I__ K__ F__； G02 X（U）__ Z（W）__ R__ F__； G03 X（U）__ Z（W）__ R__ F__；



Example：from A to B.

Absolute program: G02 X40 Z-20 I25 K0

Relative program: G02 U20 W-20 I25 K0

R program: G02 X40 Z-20 R25

G02 U20 W-20 R25

（4）Screw thread (G32)

Format：G32 Z（W）- X（U）- F（I）- SP-

G32 is the spiral interpolation machining command. Z(W) is the length of Z axis, it cuts a straight parallel thread; X(U) is the length of the X axis, and cuts a face thread; F is metric lead, range is 0.1-1300mm; I is imperial lead, range is 1-99 teeth/inch; L is the number of threads in a multi-start thread, range is 1-999, default value is 1.

The use of thread machining command is conditional that machine is equipped

with a photoelectric encoder on the spindle; otherwise the system will be placed in a standby status. When the spindle rotates clockwise, it machines a right-hand thread in the Z axis negative direction, and left-hand thread in the Z axis positive direction. The cutting feed speed F=KxN is appropriate to the machining thread, but excessive speeds will destroy the thread. This system requires N≤2000n/min, F≤3000mm/min.

For example:

Straight thread: N0000 G32 W-30 K1.5 ; straight thread of length 30, lead 1.5

Metric thread: N0000 G32 W-30 I10.2 ; thread of each inch 101/2

Face thread: N0000 G32 U-50 K2 ; thread of length 50, lead 2

If rough and fine machining is needed, add fine machining rotating speed into the thread fine machining command.

For example 1:

N0000 T0101 S1200 ; T1 is tool for rough machining, rotating speed is 1200

N0010 G00 U-1 ; advance of tool

N0020 G32 W-30 K1.5 ; rough machining

For example 2:

G00 U-62.0

G32 W-74.5 F4.0

G00 U62

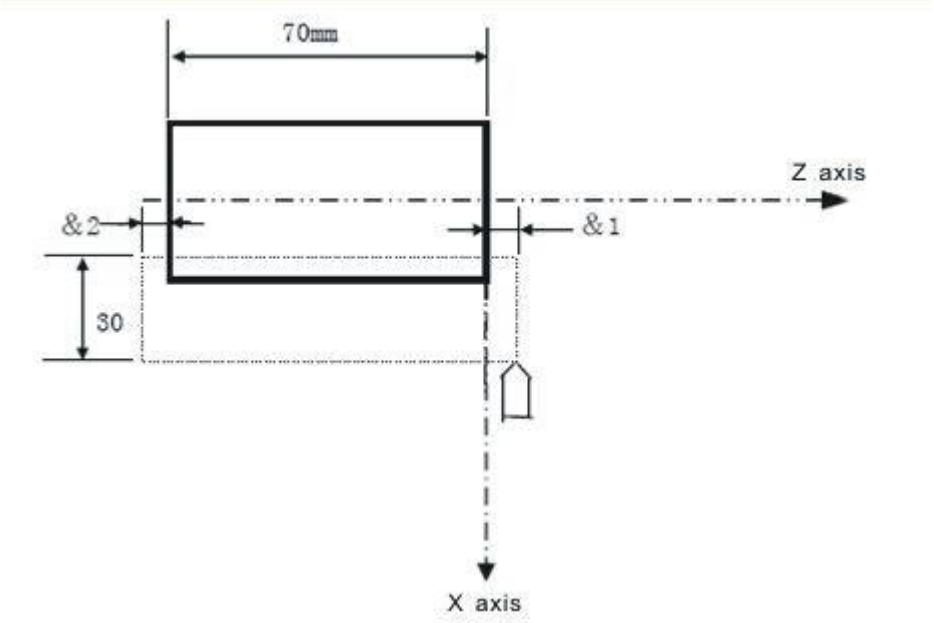
W74.5

U-64

G32 W-74.5

G00 U64.0

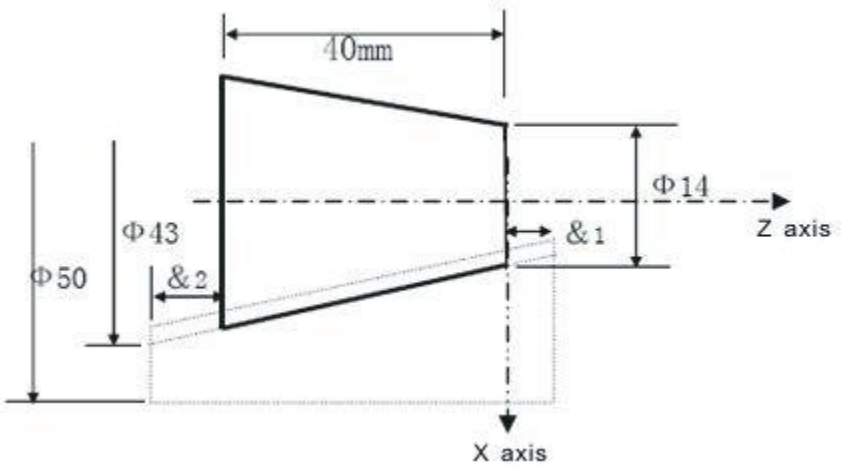
W74.5



For example 3:
G00 X12 Z3.0
G32 X41.0 Z-41.5 F3.5
G00 X50
Z3
X10
G32 X39 Z-41.5
G00 X50
Z3

（5） Tapered screw thread (G332、G333) Format：

G332/G333 Z（W）- X（U）- R - F（I）- SP-



Use the same method referred to in the G02, G03 and G32 commands.

（6）Delay (Dwell) (G04)

Delay some time before executing another command

Format：G04 P_ X_ U_

P word unit ms, means delay time。 X word unit S, means delay time。 U word unit S, means delay time。 For example：

G04 X1； delay 1s.

G04 P1000； delay 1s.

G04 U1； delay 1s.

（7）Return Reference (G28/G281/G282/G283/G284)

Return Reference command means tool go to reference point according to appointed axis.

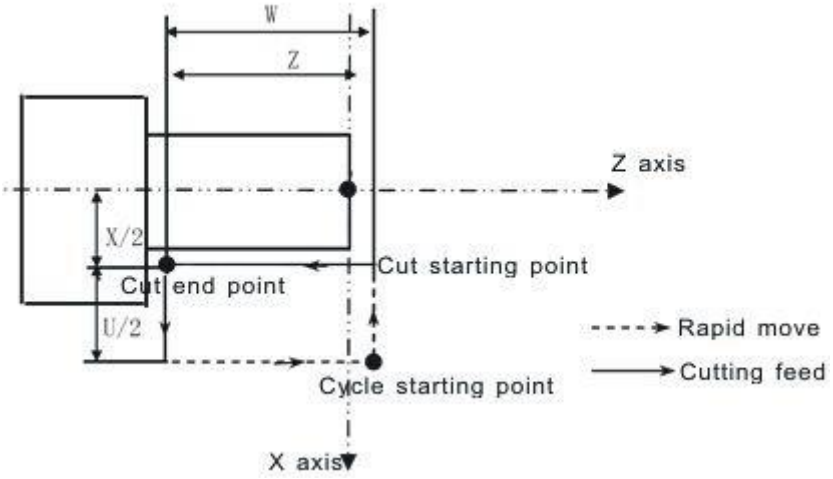
format：	G28 X/UY/VZ/W ;ZXY return to reference point
	G281 ;only X return to reference point
	G282 ;only Y return to reference point
	G283 ;only Z return to reference point
	G284 ;only A return to reference point

（8）Setup workpiece coordinate system (G50)

Format： G50 X（x） Z（z）

（9）Column or taper loop (G90)

Column loop format: G90 X (U)__ Z（W）__ F__



Column loop cutting

For example:

N10 T0101

N20 G00 X55 Z4M03

N30 G01 Z2 F100 M08

N40 G90 X45 Z-25

N50 X40

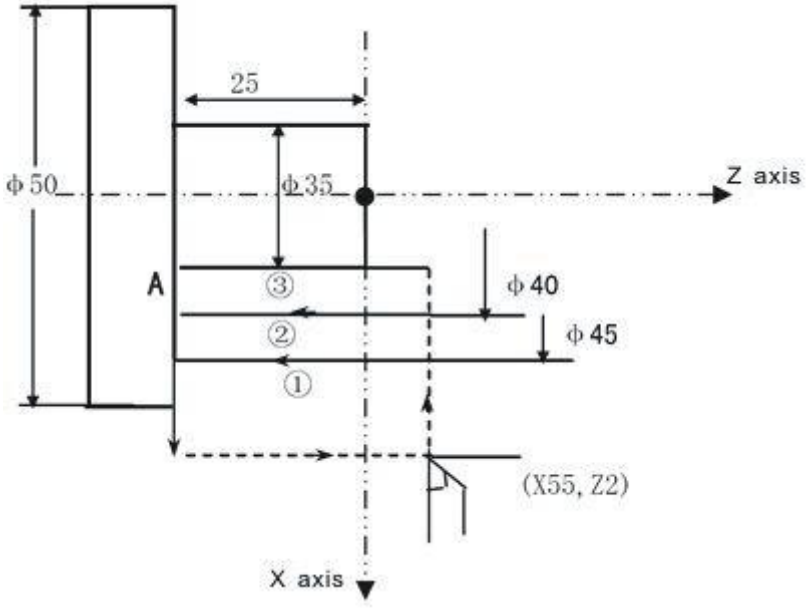
N60 X35

N70 G00 X100 Z100

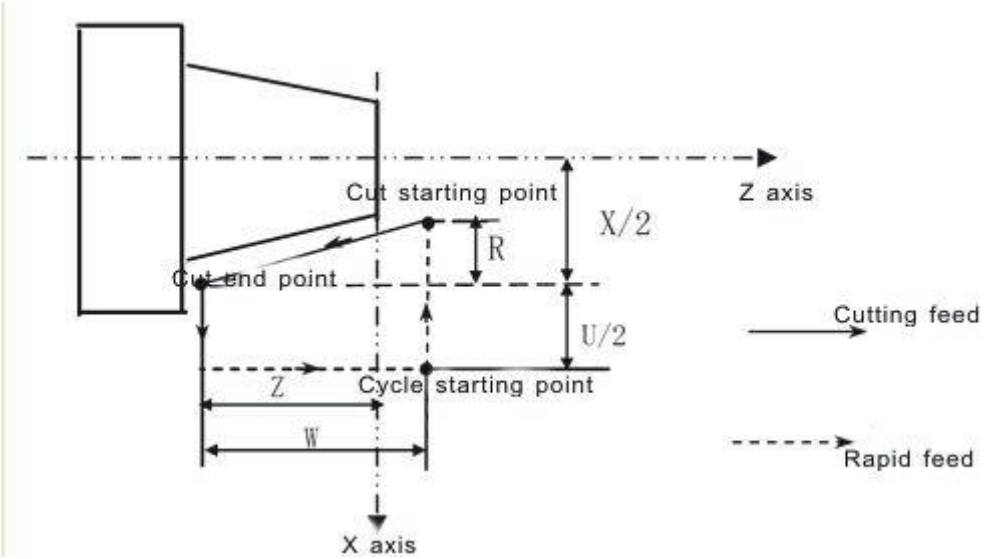
N80 T0100 M09

N90 M05

N100 M30



Taper loop format: G90 X(U)__ Z（W）__ R__ F__



Taper Loop Cutting

For example:

N10 M03 S1000

N20 T0101

N30 G00 X65 Z5

N50 G96 S120

N60 G99 G01 Z2 F1 M08

N70 G90 X60 Z-35 R-5 F0.2

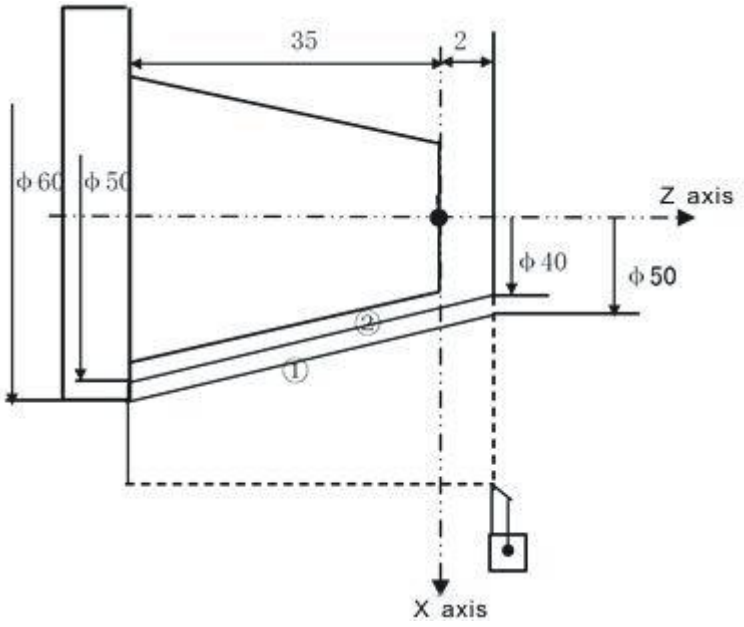
N80 X50

N90 G00 G98 X100 Z100 M09

N100 G97 S1000 T0100

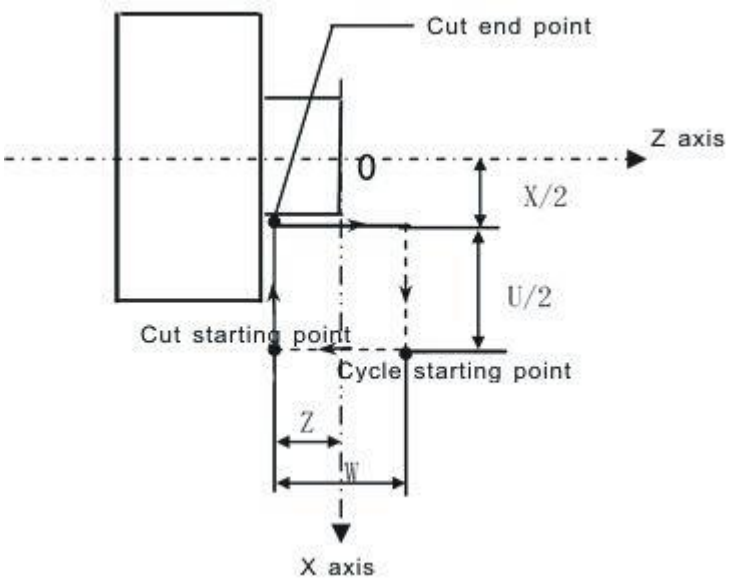
N110 M05

N120 M30



（10）End face loop (G94)

Format：G94 X (U)__ Z（W）__ F__



End face loop cutting

For example：

N10 M03 S1000

N20 T0101

N30 G00 X85 Z10 M08

N40 G01 Z5 F200

N50 G94 X30 Z-5 F100

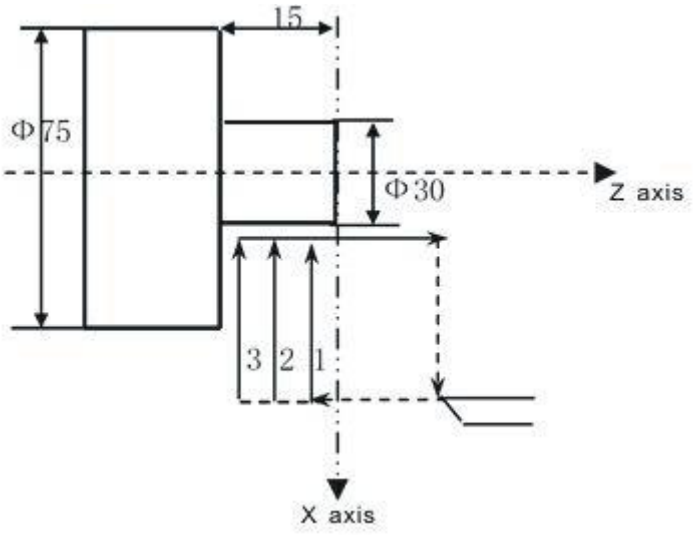
N60 Z－1

N70 Z－15

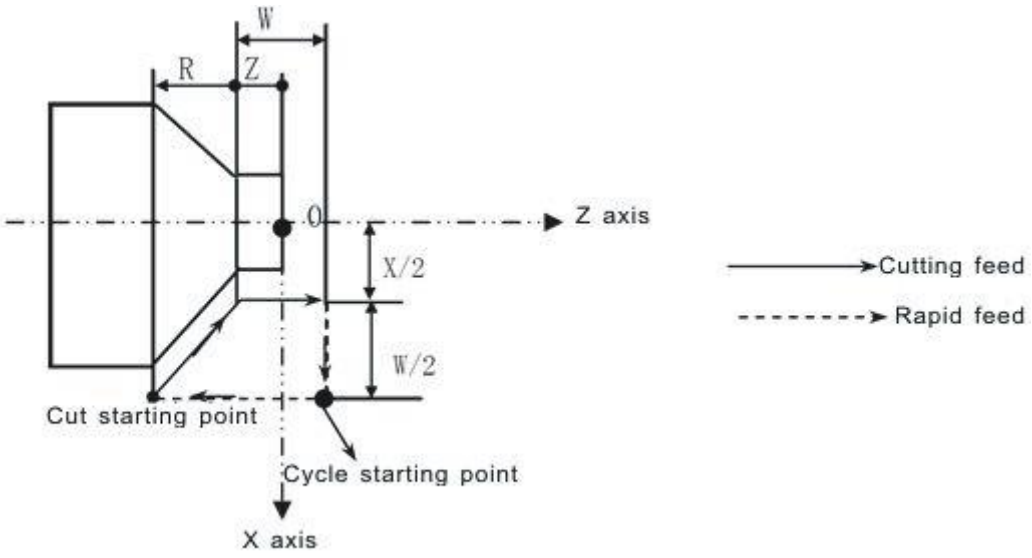
N80 G00 X100 Z60 M09

N90 T0100 M05

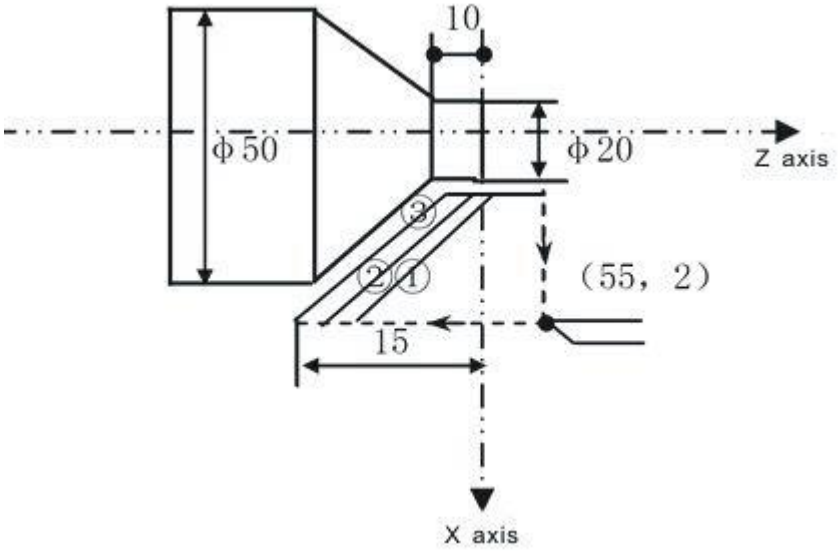
N100 M30



Taper end face loop format: G94 X (U)__ Z（W）__ R__ F__



Taper end face loop cutting



For example：

……

N40 G01 X55 Z2 F200

N50 G94 X20 Z0 R-5 F100

N60 Z-5

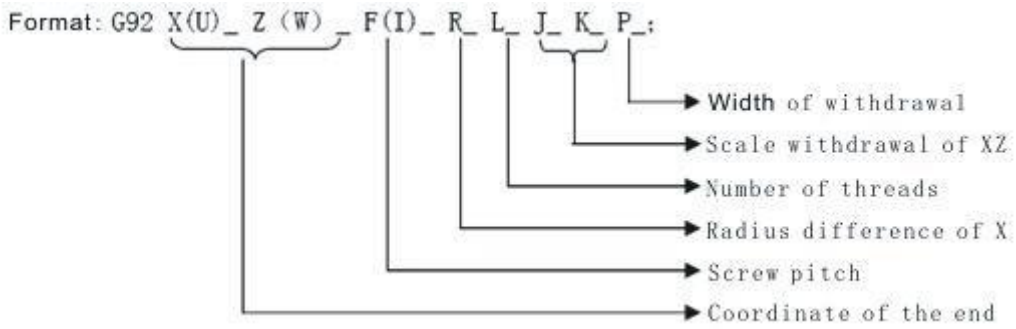
N70 Z-10

N80 G00 X Z

……

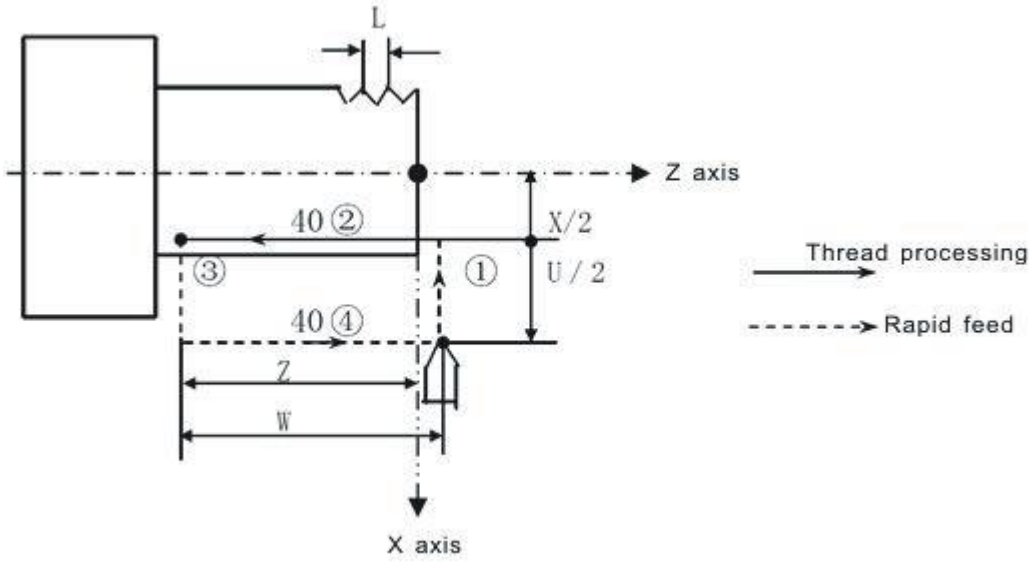
（11）Screw thread loop (G92)

Format:

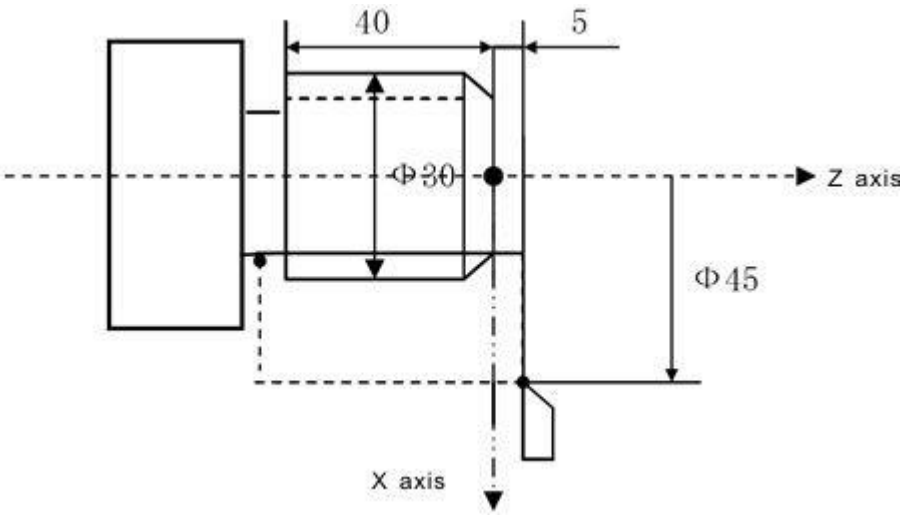


1）Straight screw thread loop format：

G92 X (U)__ Z（W）__ F/I__



Straight screw thread loop



Format:

N10 M03 S××

N20 T0101

N30 G00 X45 Z5

N40 G92 X29.2 Z-45 F1.5

N50 X28.6

N60 X28.2

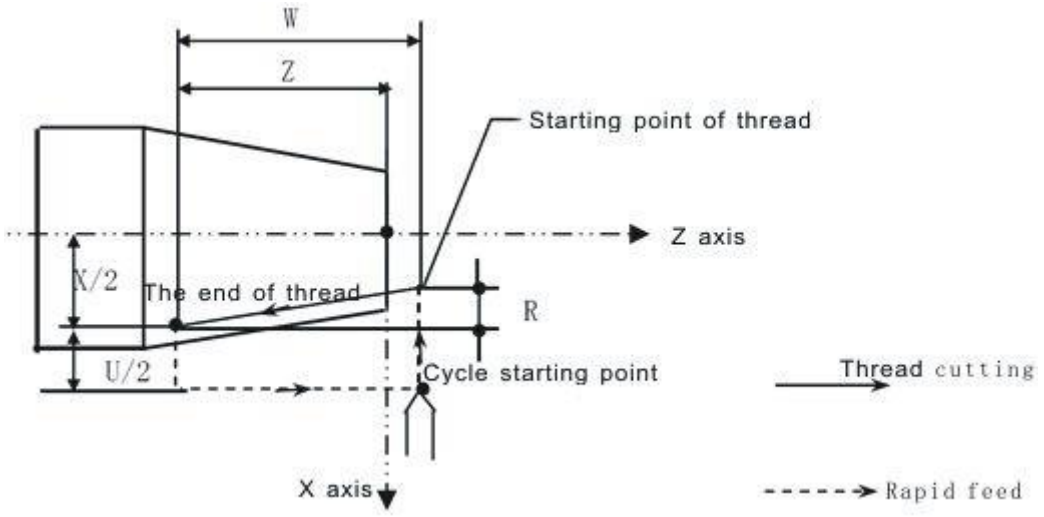
N70 X28.04

N80 G00 X100 Z50

N90 T0100 M05

N100 M30

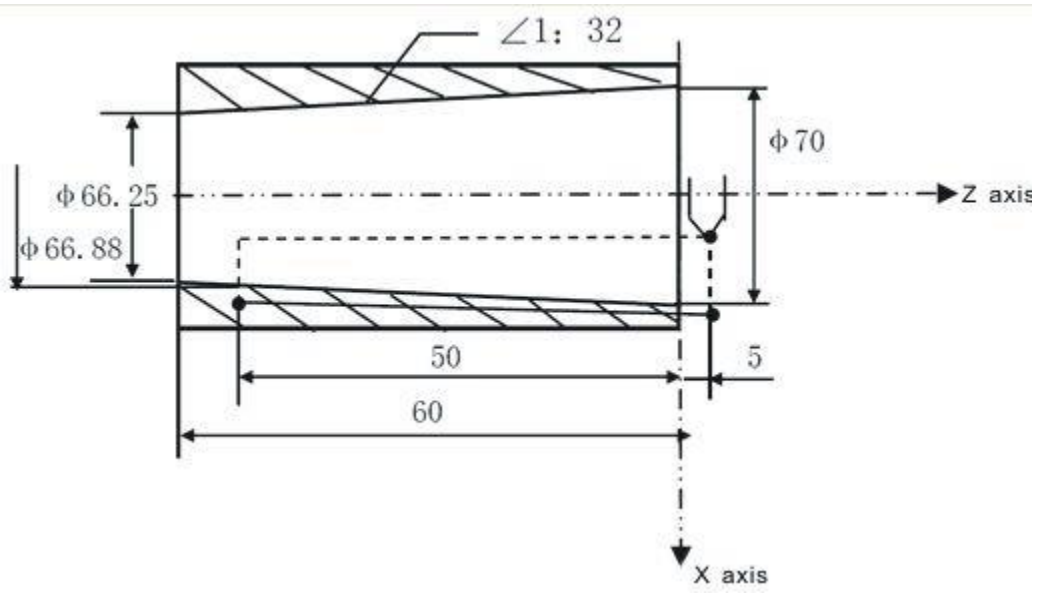
2）Taper screw thread loop format： G92 X (U)__ Z（W）__ R__ F/I__



Taper screw thread loop

For example:

```
N10 M03 S××
N20 T0101
N30 G00 X55Z10
N40 G01 X60 Z5 F100
N50 G90 X66.25 Z-60 R1.875
N60 G92 X66.88 Z-50 R1.4 I11
N70 X66.9 I11
N80 X67 I11
N90 X67.4 I11
N100 X67.6 I11
N110 X67.8 I11
N120 G00 X100 Z50
N130 T0100 M05
N140 M30
```



（12）Tapping loop (G93)

Format: G93 Z（W） F/I

For example:

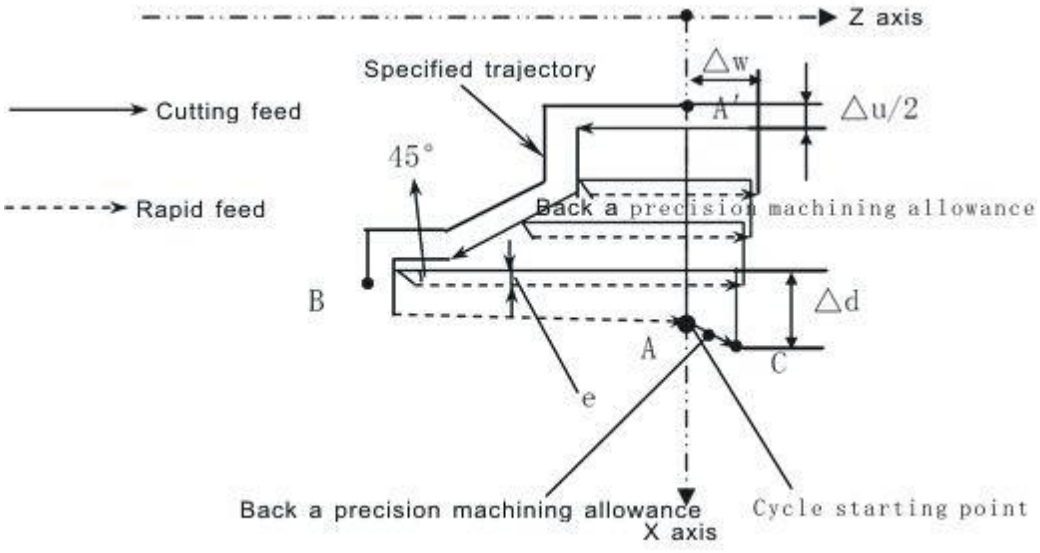
M03 S500

G04 X1

G93 Z-100 F5

G00 X50

（13）Column thick loop (G71)



Column thick loop

Format：

G71 U（△d） R(e);

G71 P（ns） Q（nf） U（△u） W（△w） F（f） S（s） T（t）;

△d: feed thickness，no signal; User parameter P1.

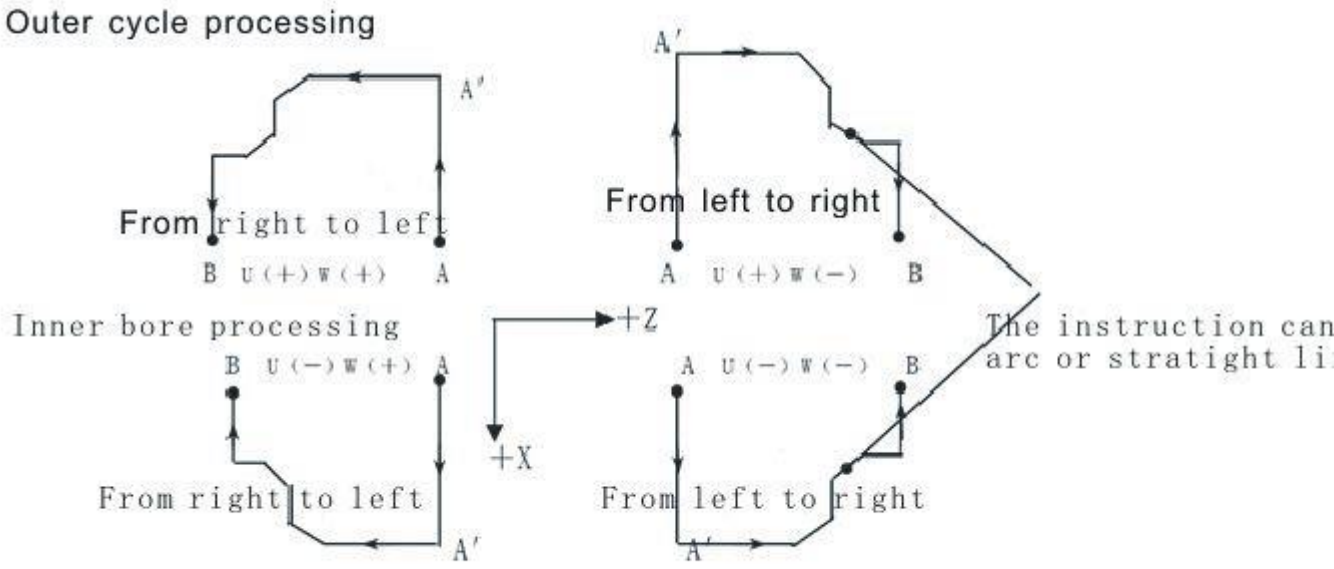
e：backward distance; User parameter P2.

ns：first N line.

nf：end N line.

△u：X remain; User parameter P4.

△w：Z remain; User parameter P5.



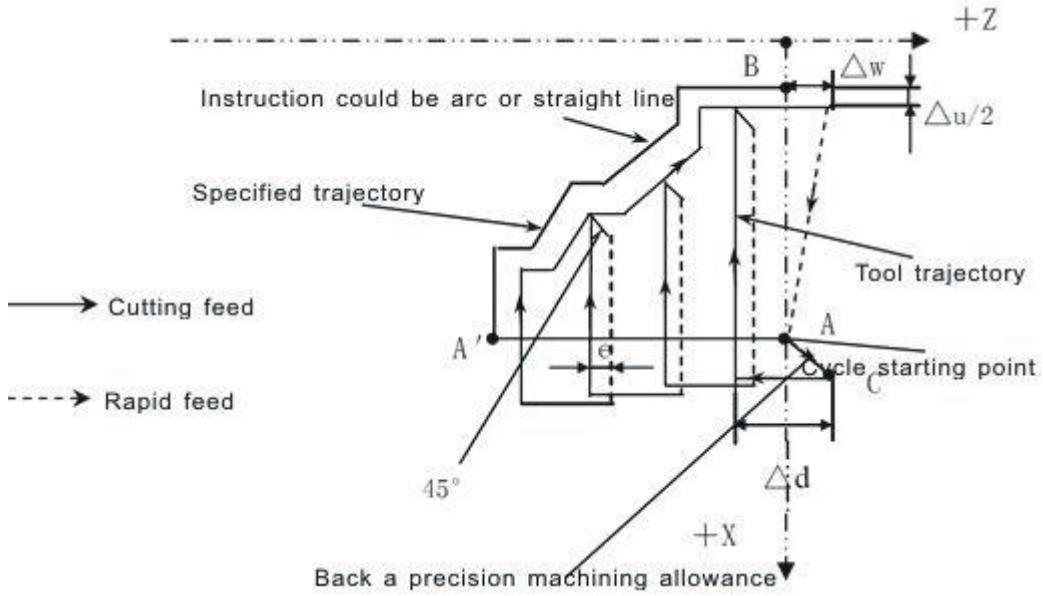
（14）End face thick loop (G72)

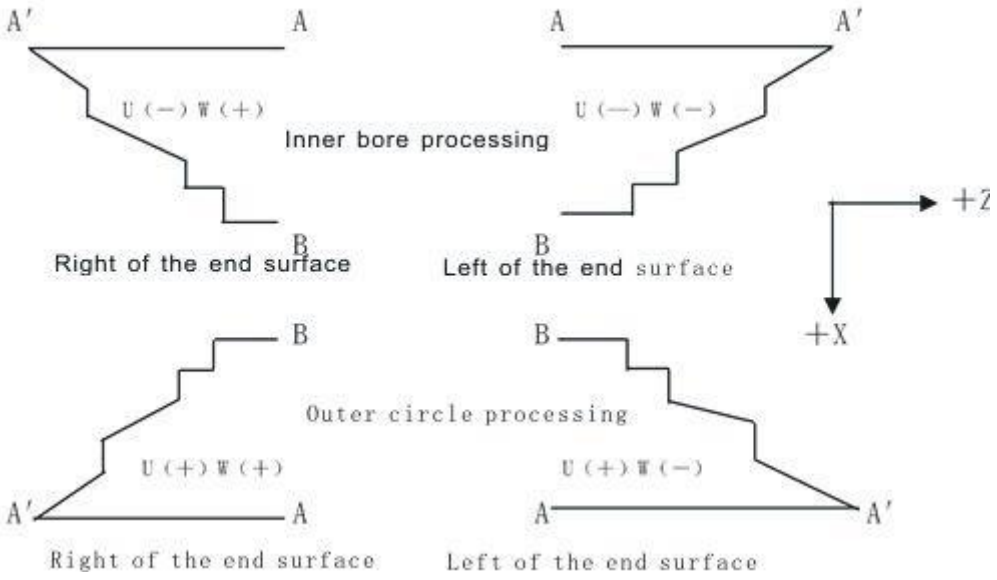
Format:

G72 W（△d） R（e）;

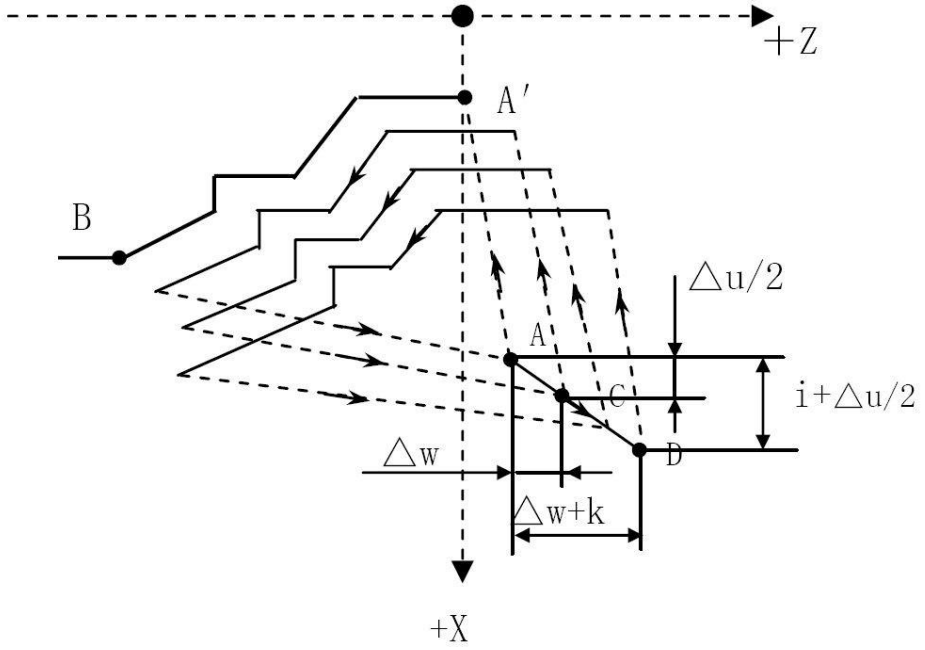
G72 P（ns） Q（ns） U（△u） W（△w） F（f） S（s） T（t）;

△d、e、ns、nf、△u、△w、f、s、t is same as G71.


+Z
B
△w
△u/2
Instruction could be arc or straight line
Specified trajectory
Tool trajectory
Cutting feed
Rapid feed
A′
A
e
Cycle starting point
C
45°
△d
+X
Back a precision machining allowance


A′
A
U(−)W(+)
B
Right of the end surface
A
A′
U(−)W(−)
B
Left of the end surface
Inner bore processing
+Z
+X
B
U(+)W(+)
A′
A
Right of the end surface
B
U(+)W(−)
A
A′
Left of the end surface
Outer circle processing

（15）Close cutting loop (G73)



Format:

G73 U（i）__W（k）__R（d）__;

G73 P（ns）__Q（nf）__U（△u）__W（△w）__F（f）__S（s）__T（t）

__;

N（ns）……; --\

……; |

. > A→A′→B ,ns to nf

. |

. |

. |

N（nf）……; --/

i：X rough thickness; User parameter P7.

k：Z rough thickness; User parameter P8.

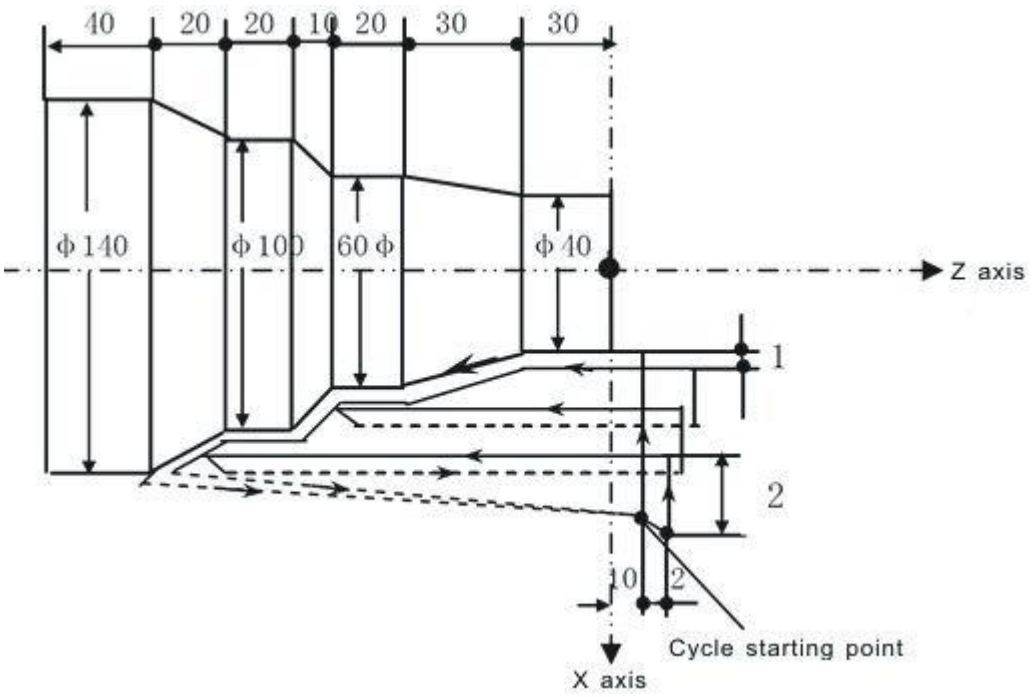
d：cutting times; User parameter P6.

Others same as G71.

（16）Finish machining loop (G70)

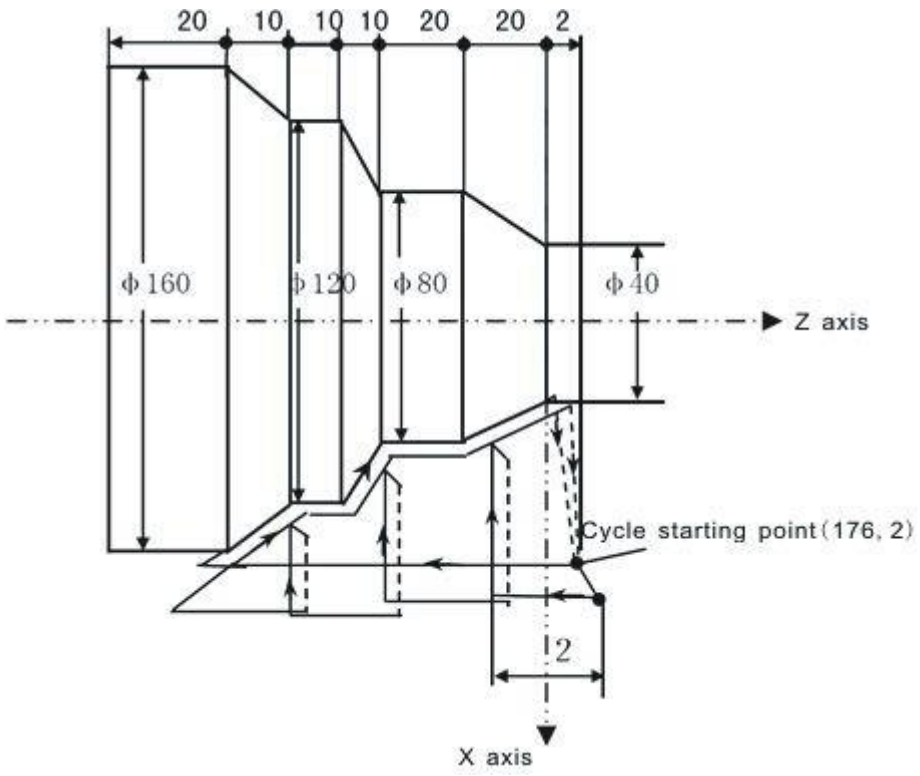
Format:

G70 P (ns） Q（nf） For example1：G71 G70



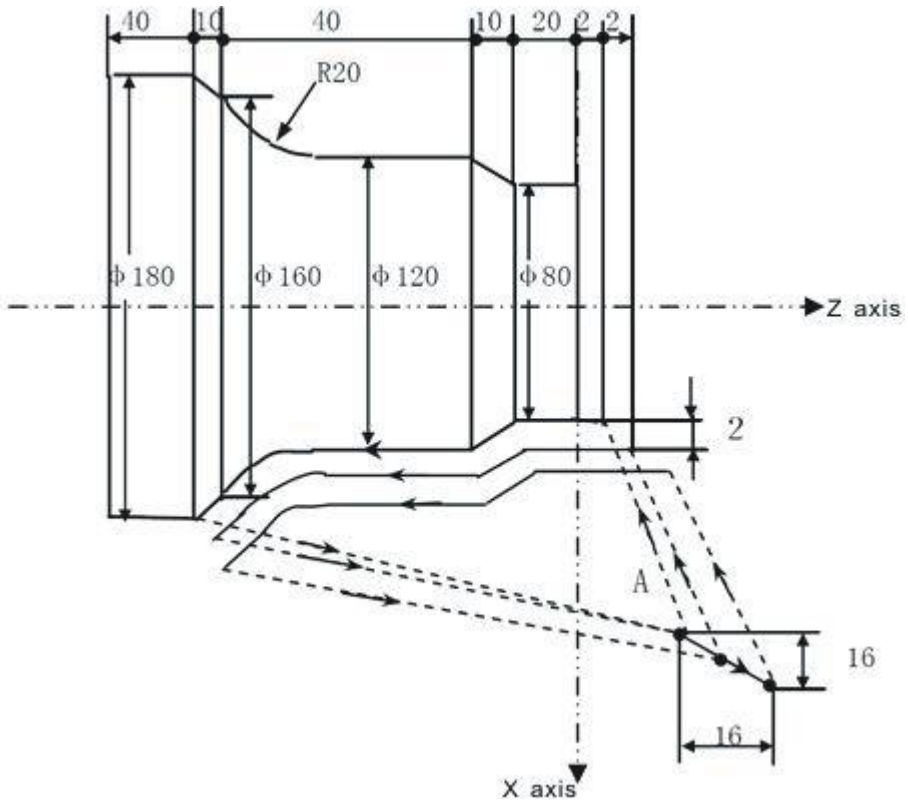
N10 M03 S1500
N20 T0101
N30 G00 X160 Z10
N40 G71 U2 R1
N50 G71 P60 Q120 U2 W1 F100 S2000
N60 G00 X40
N70 G01 Z-30 F80
N80 X60 W-30
N90 W-20
N100 X100 W-10
N110 W-20
N120 X140 W-20
N130 G70 P60 Q120
N140 G00 X200 Z50
N150 T0100 M05
N160 M30

For example2：G72 G70



N10 M03 S2000
N20 T0202
N30 G00 X176 Z2
N40 G72 W2 R1
N50 G72 P60 Q120 U2 W1 F100
N60 G00 Z-72
N70 G01 X160 Z-70 F80
N80 X120 W10
N90 W10
N100 X80 W10
N110 W20
N120 X36 W22.08
N130 G70 P60 Q120
N140 G00 X200 Z50
N150 T0200 M05
N160 M30

For example3: G73 G70



N10 M03 S3000
N20 T0303
N30 G00 X220 Z40
N40 G73 U14 W14 R0.010
N50 G73 P60 Q110 U 4 W2 F100
N60 G00 X80 Z2
N70 G01 Z-20 F80
N80 X120 W-10
N90 W-20
N100 G02 X160 W-20 R20
N110 G01 X180 W-10
N120 G70 P60 Q110
N130 G00 X250 Z50
N140 T0300 M05
N150 M30

（17）End face deep hole loop (G74)

Format：

G74 R（e）；

G74 X（u）P（△i）Z（w） Q（△k） F（f）；

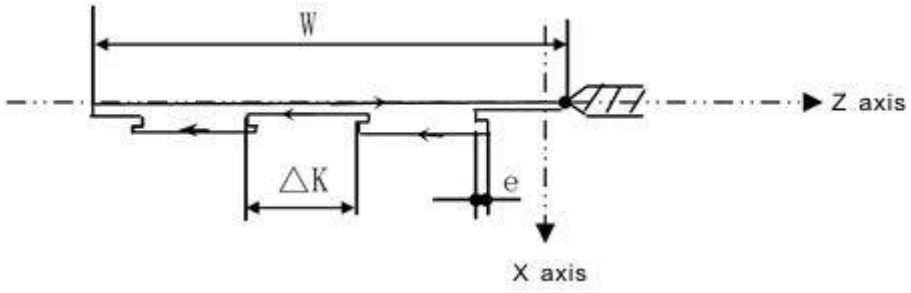
e: backward distance; User parameter P10.

Z（w） Z depth.

X（u） X end-point coordinate.

Δk： Z feed thickness; User parameter P9.

Δi： X feed thickness.



For example:

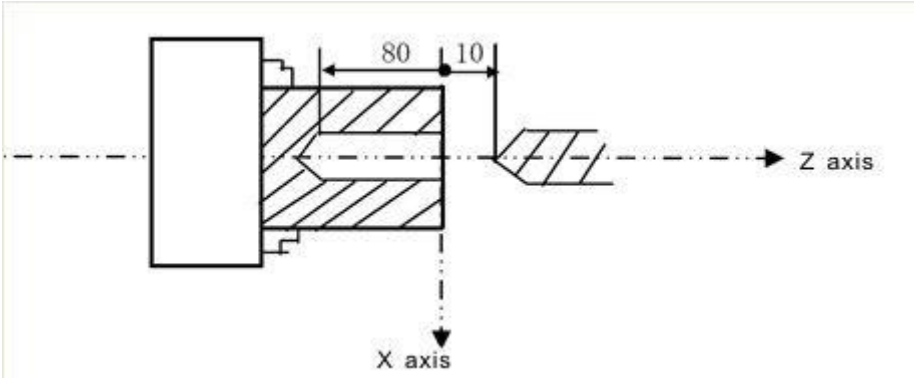
N10 G00 X0 Z10

N20 G74 R2

N30 G74 Z-80 Q10000 F800

N40 G00 X50 Z50

N50 M30



（18）Slot cutting loop (G75)

Format： G75 R(e)__;

G75 X(U)__ P(Δi)__ Z（w）__ Q（△k）__F(f)__;

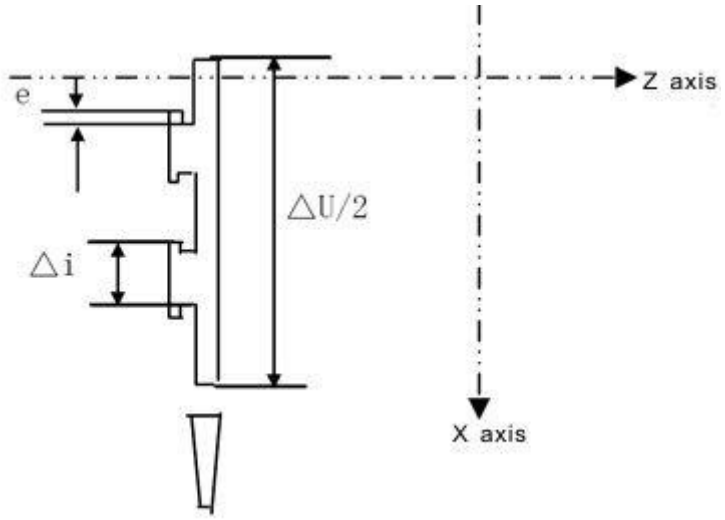
e: backward distance; User parameter P10.

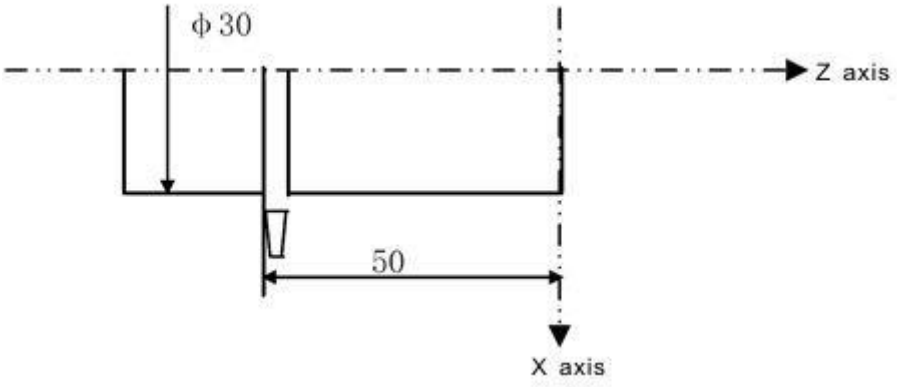
X（u） X depth.

Z（w） Z end-point coordinate.

Δi： X feed thickness; User parameter P9.

Δk： Z feed thickness.

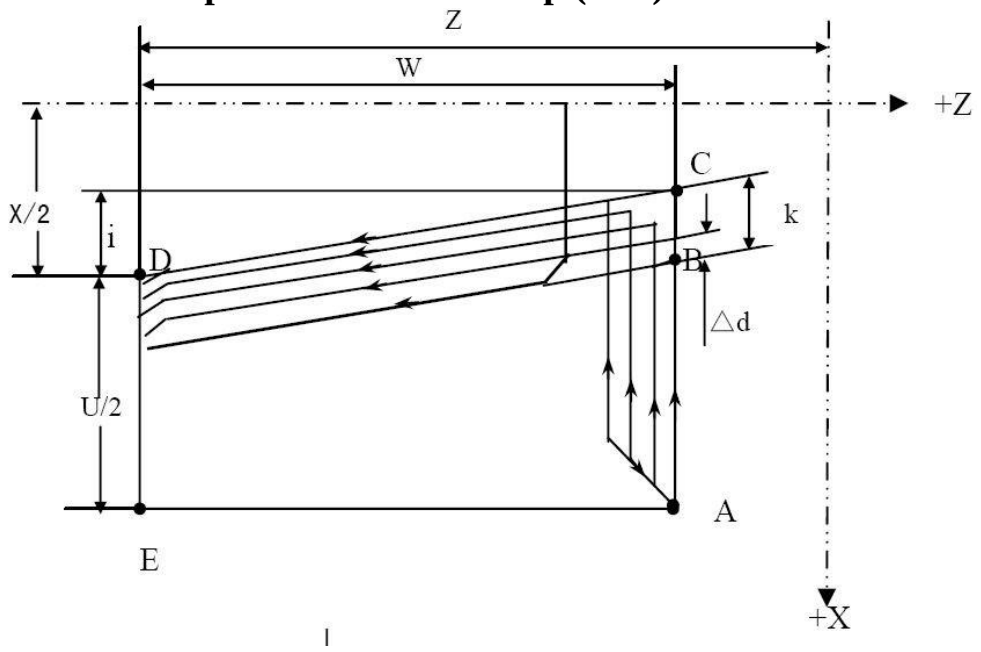


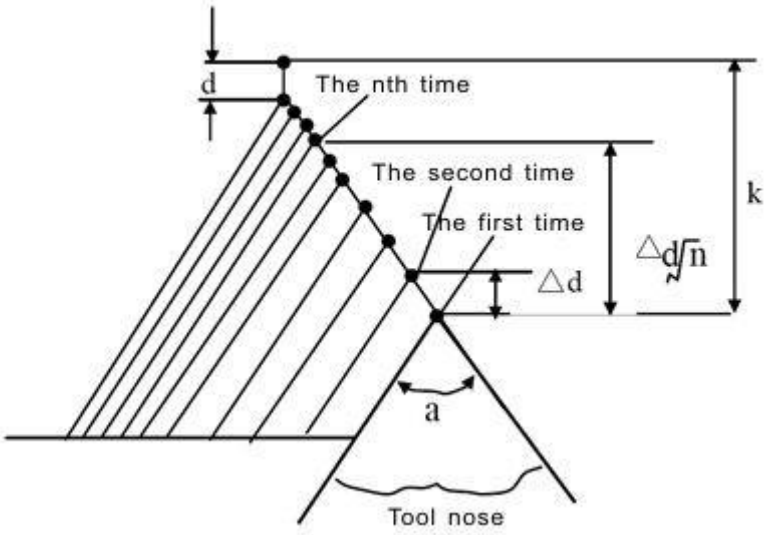


For example:

N10 M03 S1000
N20 T0101
N30 G00 X35 Z-50
N40 G75 R1
N50 G75 X-1 P5000 F60
N60 G00 X100 Z50 M09
N70 M05
N80 T0100
N90 M30

（19）Complex screw thread loop (G76)





Cutting method:

Format:

G76 P(b)(c)(m)(r)(a) Q(Δd min) R(d) ;

G76 X(U) Z(W) R(i) P(k) Q(Δd) F(f) L(L)[or SP];

b：0———degression feed 1———equidistant feed

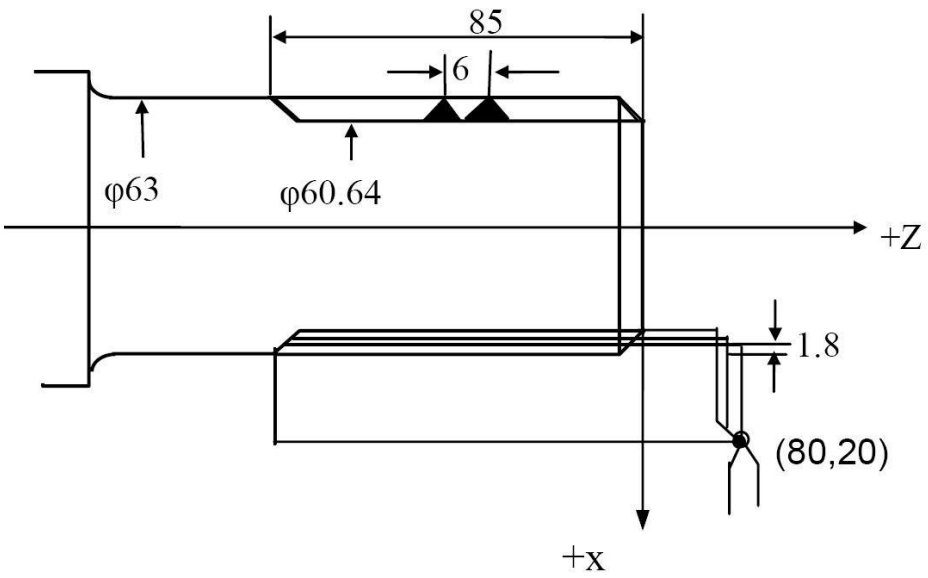
c：0———right enter 1———left enter

m: finish turn times, User parameter P11. r: quit length, User parameter P12.

a: thread tooth angle(degree), User parameter P13. Δd min: minimal cutting depth, User parameter P14. d: finish turn remaining, User parameter P15.

I：X taper screw thread feed measure. f: metric lead.

L: multiple thread head numbers. SP: start angle：0-360



N10 M03 S1000

N20 T0101

N30 G00 X80 Z20

N40 G76 P00011060 Q100 R0.1

N50 G76 X60.64 Z-85 P3680 Q1800 F6.0

N60 G00 X100 Z50

N70 T0100

N80 M05

N90 M30

（20）Loop (G22，G800)

Format：G22 L-

.

.

.

G800 ;End

For example:

N0000 M03 M08

N0001 G0 X200 Z200

N0002 G01 W-100 F300

N0003 G22 L6 ;loop 6 times

N0004 G01 U-22 F100 N0005 W－11U6 N0006 W-30

N0007 W-10 U5 N0008 G0 U10

N0009 W51

N0010 G800 ;loop end

N0011 G26

N0012 M30

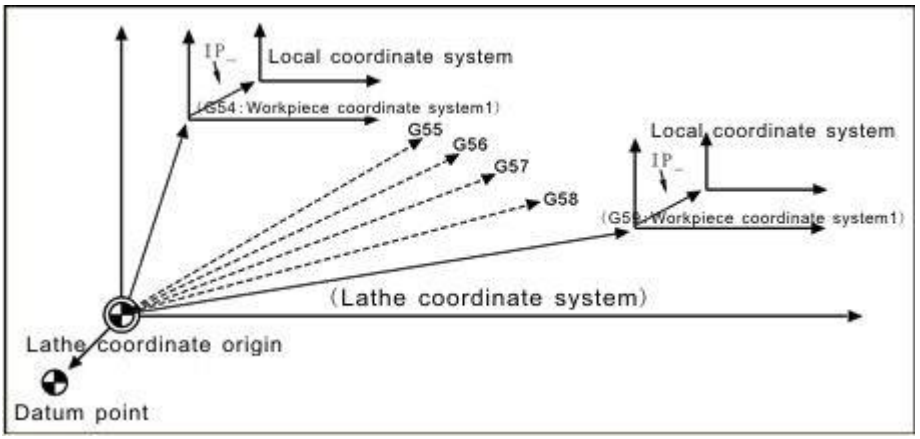
（21）Part coordinate setup (G52)，Setup tool co-or（G184、G185）

Format:

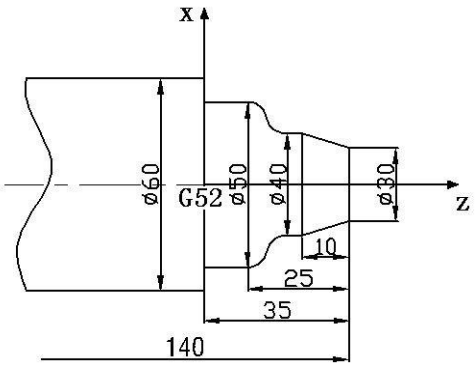
G52 X_Z_; absolute coordinate setup G52 U_W_; relative coordinate setup G52 X0 Z0 ; cancel

G184 X_Z_P_L_;setup current&P&L tool absolute coordinate G184 U_W_P_L_;setup current&P&L tool relative coordinate G185 X_Z_; setup all tool absolute coordinate

G185 U_W_; setup all tool relative coordinate



Part coordinate setup For example：



N1 G00 X60 Z20

N2 G52 X0 Z-236

N3 T0101

N4 M03 S800 M08

N5 G01 Z35 F100

N6 X-1

N7 X70

N8 G71 U2 R1
N9 G71 P10 Q15 U0.5 W0.5 F150
N10 G01 X30
N11 X40 Z25
N12 Z20
N13 G02 X50 Z15 R5
N14 G03 X60 Z10 R5
N15 G01 Z0
N16 G00 X70
N17 G52 X0 Z0
N18 M05
N19 M30

（22）Back start point (G26、G261、G262、G263、G264)

Format：

G26、G261、G262、G263、G264;　　XZ、X、Y、Z、A

Format:

```
N0000  G00 X120 Z300      ;
N0001  G01 X150 Z50 F160  ;
N0002  G26                ;
N0003  M2                 ;
```

(23） Back to G25 point (G25、G61、G611、G612、G613、G614)

Format:

G25 ; Save current coordinate G61, G611, G612, G613, G614; XZ, X, Y, Z,A

For example:

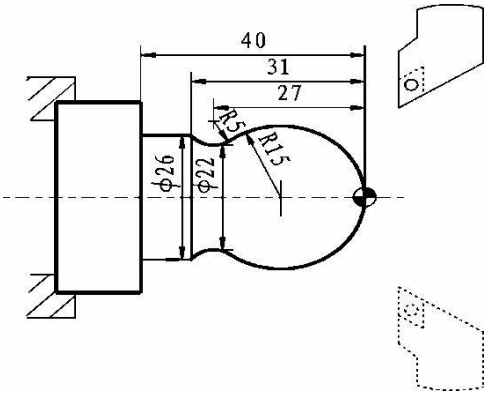
N0000 G0 X20　　Z80
N0001 G01 U5 W－16 F200
N0002 W－100
N0003 G00 U10
N0004 Z80
N0005 G25
N0006 G01 U10 W-30
N0007 G0 X100 Z200
N0008 G61
N0009 M2

（24）Continuous feed cutting (G60/G64)

Format:　　G60　; cancel
　　G64　; continue feed

(25）Constant speed cutting (G96/G97)

Format：G96 S_ ; Constant speed cutting
G97 ; cancel



For example:
N1 T0102 X40 Z5
N2 M03 S400
N3 G96 S80
N4 G00 X0
N5 G01 Z0 F60
N6 G03 U24 W-24 R15
N7 G02 X26 Z-31 R5
N8 G01 Z-40
N9 X40 Z5
N10 G97 S300
N11 M30

（26）Feed mode (G98、G99)

Format:
G98 feed per minute command G99 feed per revolution command

（27）Check skip (G31、G311)

Format：G31 X_ Z_ F_ P_ ; Skip on No alarm G311 X_ Z_ F_ P_ ; Skip on alarm

P:N-line+(X00/X39+1000 or 2000)， 1000 means availability skip, 2000 means invalidation skip.

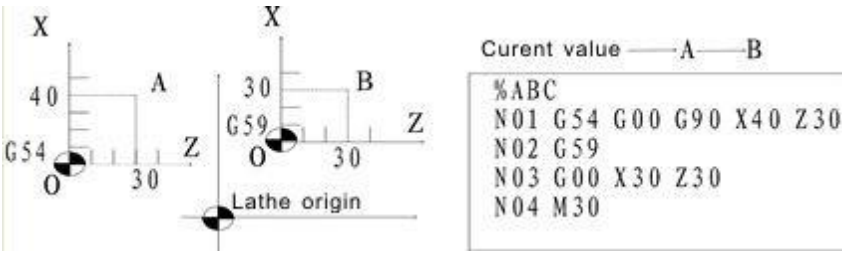
For example：G31 X50 Z100 F100 P331022 ; if X22 availability then go to N33.

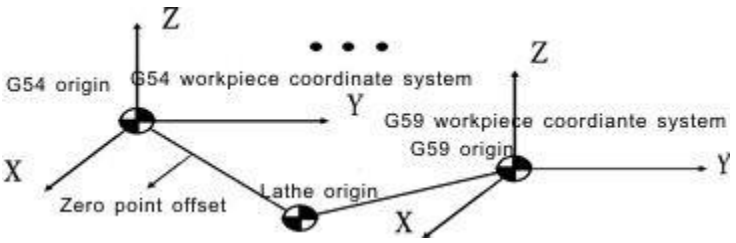
G311 X50 Z100 F100 P2021 ; if X21 invalidation then go to next line.

(28）Work coordinates (G53/G54/G55/G56/G57/G58/G59)

Format：G53/G54/G55/G56/G57/G58/G59

G53 machine coordinate
G54 workpiece coordinate1
G55 workpiece coordinate 2
G56 workpiece coordinate 3
G57 workpiece coordinate 4
G58 workpiece coordinate 5
G59 workpiece coordinate 6





(29）Tool radius compensation (G40、G41、G42)

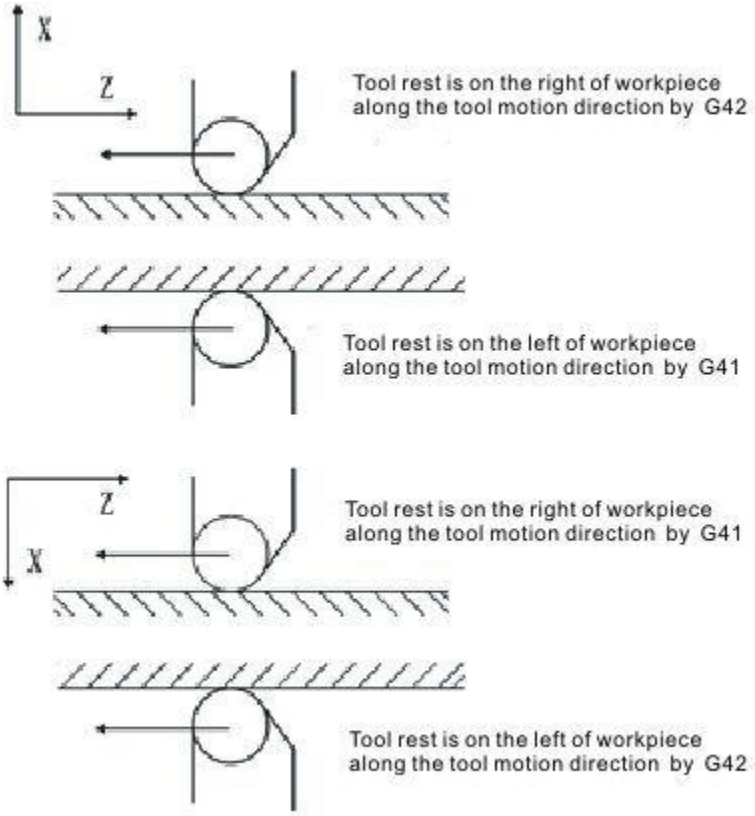
Format : G40 ; Tool radius compensation cancel.
G41; Tool radius compensation of cutter offset to the left of workpiece.
G42; Tool radius compensation of cutter offset to the right of workpiece.

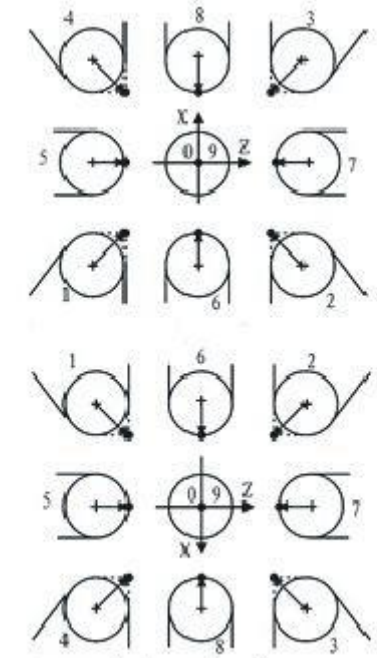
The left and right in G41, G42 are viewed from direction of cutting, where the tool lies to the left or right of workpiece. Tool radius is designated by R.

Executing offset begins at the program line of G41, G42. In the closed angle, the system generates an automatically transiting arc, allowing the tool radius offset vector of the last program line to transfer to that of the next program line. The Tool offset vector describes the value and direction method of tool offset, with the radius vector as the tool radius. For arc, its direction is in the radius direction. For line, its direction is vertical to the line direction.

When it transits at the closed angle arc, it will cause errors when the angle is less than 180 degree, because the transition becomes the inner closed angle transition. This system only transits outer closed angle, and remains valid only for G01, G02, G03, that is, the outer closed angle between strait line and strait line, arc and arc,

strait line and arc can generate transiting arc.

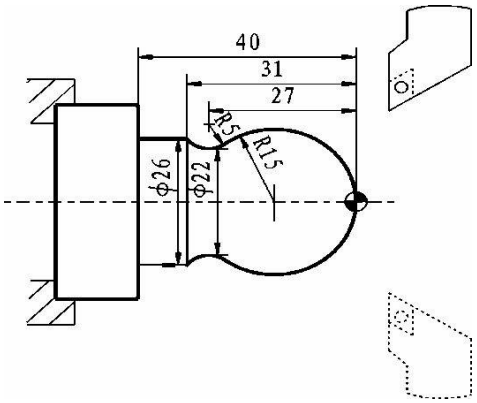




Rear Toolpost

Front Toolpost

Left and right compensation



Tool parameter For example：

N1 T0101
N2 M03 S400
N3 G00 X40 Z5
N4 G00 X0 G42
N5 G01 Z0 F60
N6 G03 U24 W-24 R15
N7 G02 X26 Z-31 R5
N8 G01 Z-40
N9 G00 X30
N10 G40 X40 Z5
N11 M30

（30） Polar coordinates program (G15/G16)

Format： G15 ; cancel
G16 IP-(XZ) ; pole coordinate

For example：

N1 G16 X0 Z0
N2 G01 X30.0Z100.0 F200
N3 X150.0
N4 X270.0
N5 G15
N6 M02

（31）Switch millimeter and inch (G20/G21)

Format：	G20; Inch; G21; Millimeter;

（32）Call sub-program (M97/M98/M99)

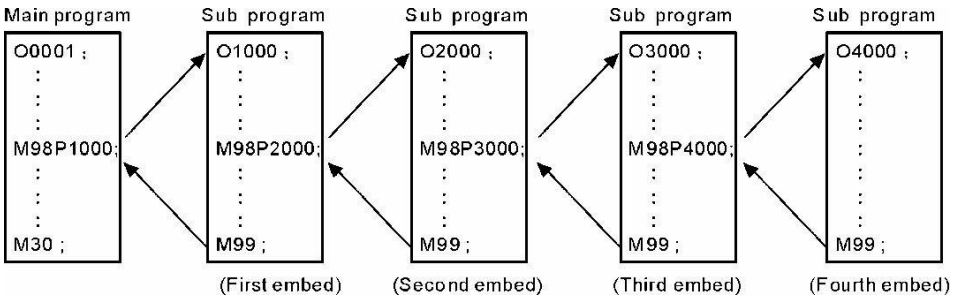
M97 Psub; Non-condition jump to Psub sub-program

M98 Psub Lnnn; Call sub-program。P word points to the name of the sub-program.

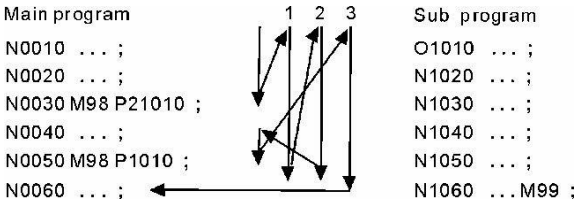
For example： Psub\\%ab12 means the name of sub-program is CNC\\sub\\%ab12 ， L word point out call times。

M99 Return from sub-program

The Sub-program can embed other sub-program calls as follows:



For example



（33）T, Tool

Format function

Tab a: Tool number, b: compensate number Format:

N0000 T0101

N0001 G00 X30 Z500

N0002 T0303

N0003 G00 X50

N0004 T0505

N0005 M02

（34）S、SS Spindle speeds

The Primary Spindle use “S”, with the speed parameter P36 controlling the maximum rotational speed, output as a 0-10V frequency conversion voltage.

The Secondary Spindle uses “SS”, and speed parameter P40 to control the maximum secondary spindle speed, output as a 0-10V frequency conversion voltage.

（35）F, Feed rate

F is used for G01, G02, G03 etc. feed rates.

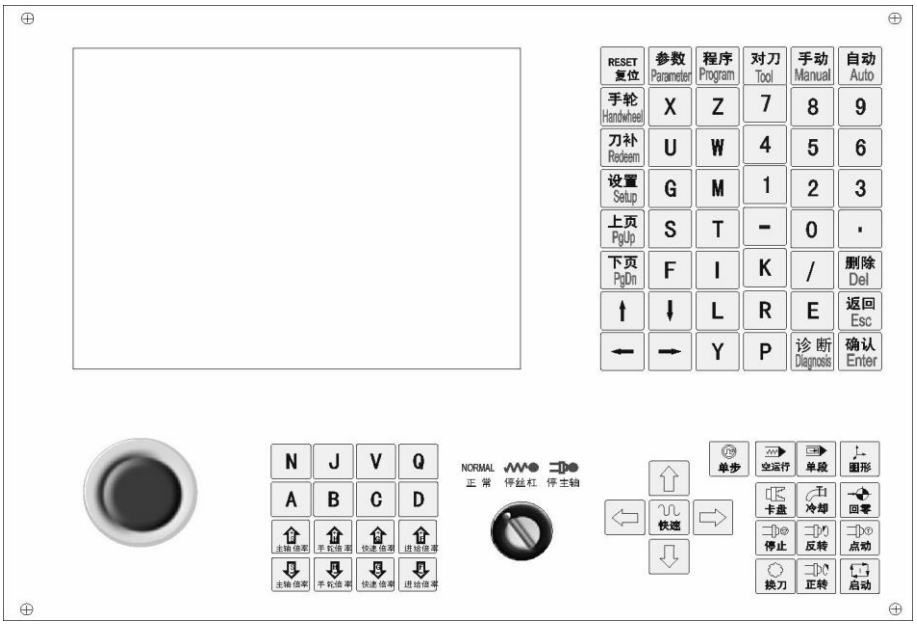
If G98 (fpm) is active: F0.01-20000mm/min, if G99 (fpr) active: 0.001-500mm/r.

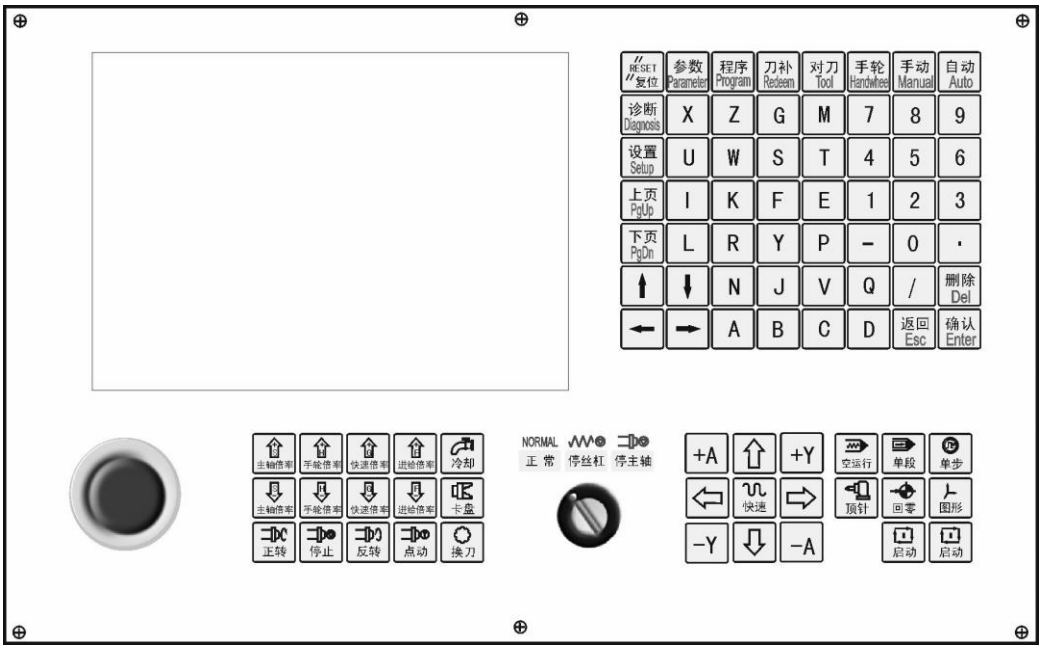
Chapter 3 Operation explanation

3.1 Summary

When using the NC system, first master the parameters of the system, program editing, manual operation, then auto operation. Then you can operate easily. Here are some details to assist you.

3.2 Operation panel and switches (Two different panel Styles)





3.3 Keyboard operations

3.3.1 Rate increase

(1) Rapid over-ride (G) 

There are six steps of rapid override adjustment, ranging from 5% to 100%. By adjusting the Rapid Over-ride key, the G00, G26, G28, G611, G613 commands rapid feed in fixed cycle, and rapid manual feed are affected.

(2) Feed over-ride (F) 

There are sixteen steps of Feed Over-ride adjustment, ranging from 0% to 150%. By adjusting the feed override key, the G01, G02, G03 commands, and the feed rates in both the fixed cycle and manual operation modes are affected.

(3) Spindle over-ride (S) 

There are sixteen steps of spindle override adjustment, ranging from 5% to 150%. By adjusting the spindle override key, the speed of the primary spindle is affected.

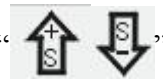
3.3.2 Switches

Switch	Functions
	**Emergency Stop**. All Drivers and motors stop immediately. The spindle and coolant stop, and waits for the release of the button. All values are initialized.
	**Intervention switch**: When a program is running, or in manual state, this can be used to make a real-time adjustment of feed speed; Spindle percentage switch: When the spindle is running, this can be used to adjust the speed accordingly.

Buttons:

Keyboards	Functions
Letter and Number keys	A B C S E F G H I J K L M N O P Q R S T U V W X Y Z 1 2 3 4 5 6 7 8 9 . -: For program instructions, parameters, editing; Number keys are used for data input and selecting sub-menus.
Edit keys	"↑, ↓, →, ←, Del, PgUp, PgDn" for programming. Direction keys can be used for selecting menus.

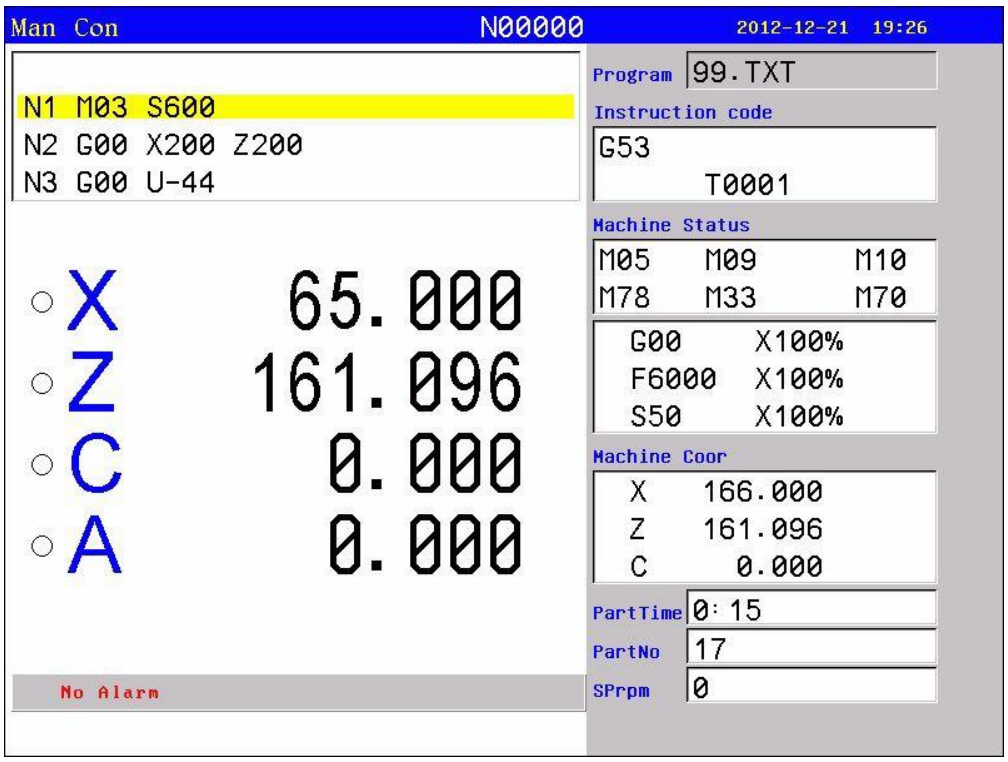
Function key	"Esc" Return to upper level or stop a operation "Enter" Select a sub-menu and changing a newline "Del" Delete program "program" Entering or editing a program "parameter" Enter parameter setting "diagnosis" Enter the diagnosis I/O function "manual" Change control to Manual status "handwheel" To use hand-wheel function "Tool" For confirming current tool position in machine coordinate system. "Redeem" For amending tool change errors "Auto" Change control to Automatic status "MDI" Change to MDI function  " Select auto-coordinates/diagram machining  " Select single line or constant work '  " Select coordinate mode or diagram mode speed simulation  " Select manual increment or manual continuous

Control key	“  ”spindle Cw, CCw rotation “  ” Coolant on/off “  ” Shift between electric tool carrier and gang tool carrier “  ” Shift between hand-driven continuous high speed and low speed. “  ” Return all axes to Datum Point (Home) “  ” Spindle Chuck close/open “  ” Tailstock extend/retract “  ” Hand-wheel gear ratio selection “  ” Adjust spindle speed “  ” Adjust feed rate “  ” Rapid (G00) over-ride	
Feed key	 +Y +A–Y -A	For X, Y, Z, & A axes direction feed

3.4 Manual operation

The system is based on intuitive and convenient one-level menu operations, and comprehensive information prompts.

First Powering On the system, the Manual Interface is displayed. (See below.)



3.4.1 The key of manual operation

(1) “F”: Taking mm/min as the unit to set the manual feed speed, the input range is from 1 to 30000mm/min. And the input method according to data input method in parameter.

(2) “ ”: Switching cycle from “manual continuous” to “manual incremental”

(3) “S”: Set the speed of the primary spindle. The range is from 0 to 99999, the maximum speed is set by Speed Parameter No.36.

(4) “I” Modify the increment in Manual Step Increment.

(5) “ ”： Press once to increase or decrease the Feed Rate by 10%. If Axis Parameter No.1 is **0**, the range is from 0% to 150%, in 16 steps.

(6) “ ”: Press once to increase or decrease G00 or Manual Rapid override by 20%. The range is from 5% to 100%, in 16 steps.

(7) “ ”: Press once to increase or decrease the Spindle override speed by 10%. When Axis Parameter No.2 is **0,** the range is from 5% to 150%, in 16 steps.

(8) “ ”: To switch between increments of “0.001” “0.01” “0.1” in the handwheel function.

(9) “Diagnosis”: Enter the diagnosis display for inputs and outputs.

(10) “Setup”: To set a G54 to G59 value in workpiece Grid Shift coordinates. (Use “MDI” to set G54 to G59 values in Lathe Coordinate (G53).

(11) “Auto”: Select automatic mode.

(12) “Manual”: Select manual mode.

(13) Spindle control:  ” Controls the spindle for clockwise, counterclockwise, stop, or commands M03, M04, M05.

When Axis Parameter No.56 is set to **8**, pressing the “spindle on counterclockwise” button “Jogs” the spindle counterclockwise.

(14) “Cooling”: Coolant on or off correspond to commands M08, M09. (15) “Chuck”: Chuck clamp or unclamp corresponding to commands M10, M11.

(16) “Manual speed controlled”: Press “1” “2” “3” “4” “5” “6” “7” “8” “9” to set feed override “F30” “F60” “F120” “F250” “F500” “F1000” “F1500” “F2000” “F2500” “F3000”.

(17) “Tailstock”: Tailstock extend or retract corresponds to commands M79, M78.

(18) “Switch manual continuous or increment”: Press  for Manual Continuous or Increment. Displays I=XXXX.XXX when Manual Increment is active.

(19) “Back to datum point”: Press  and X or Z, the X or Z axis goes back to the datum point automatically; Press “0”X axis firstly and then Z axis; Press “Esc” to cancel the process. The return speed is controlled by Speed Parameters No.31 and No.33, and the direction is determined by Axis Parameter No.28.

(20) “Tool carrier controlled”: Press  to change to next tool automatically if a gang tool carrier is fitted, or if an electric tool carrier, it will stop at the next tool position, and the selected tool data will be redeemed.. If a particular tool is required, Press “T” and number to change to this tool.

(21) “Coordinates feed”: Press “↑ ↓ ← →”correspond to feeds in X and Z axes in positive or negative directions.

(22) “Switch speed”: Press  to switch the feed to a system feed rate set by Speed Parameters No.1 and No.2 when it is in Coordinate Feed. Upon releasing the button, it will revert to the previous speed. If a desired speed is higher than the speed set in the parameter, it will feed at the rate set in the speed parameter.

(23) “Switch coordinates display”: Press “PgUp” or “PgDn” to switch the display between “Relative”, “Absolute”, and “Machine” coordinates.

(24) “Part no. clear”: To Delete a selected program, press “Del” and “Enter”.

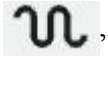
(25) “Back to G53 coordinate datum point”: Press “Q” and “Enter”.

(26) “Incremental coordinate”: Press “Setup” to fix or set to “**0”** after selecting “Relative” coordinates.

PS: Lathe coordinate clear: Press “E” in parameter and then press “Enter”.

3.4.2 Manual continuous

For Continuous feeding operations, with time as the basis, press to feed, up to stop feeding. Making sure the axis and using “↑ ↓ ← →” to feed, the speed of feed is determined by display on the interface (F) times the rate.

When Continuous starting, press  ” to switch to Speed Parameter to No.1

and No.2. If set the desired speed is higher than the speed set in the Speed Parameters, the feed speed will be the parameter speed times the rapid override setting.

To better control the cutting in a single axis, pre-set the feed speed in Manual mode first. Press "F" and input the desired feed rate.

If an axis is moved beyond the hard limit point, feeding will stop, and the operator will be prompted to reverse the axis direction to recover from the fault.

The manual maximum speed is limited to that set in Speed Parameter No.3. If a speed higher than Parameter No.3 is requested, it will not be achieved.

When Other Parameters No.38 is set to **8**,  " changes into a switch. Press once to toggle On, press again to toggle Off.

3.4.3 Manual increment

This operation is to set the value of increment as the basis, press "↑ ↓ ← →" once to run a value of increment. It will prompts "I=0010.000" in manual increment represent for the value of increment is 10mm, press "I" to revise and Enter. But also

press "handwheel" and " "to switch the value into 0.001mm 0.01mm 0.1mm. The speed is the speed on display (F) times the rate.

3.4.4 Return to lathes datum point (reference point)

There are two ways to return to the datum point in this system. This can be controlled by a switch, or by a set floating point. The method is as follows:

Switch for datum point:

A return to datum point operation consists of feeding each axis back to a lathes set datum point position in turn. When the parameter of a feeding axis is returned to datum point 0, the system controlling that axis detects the datum point and returns a pulsing signal of "Zero". The coordinate data for that axis will be reset to 0.

Every time a CNC is powered On, a major Alarm is reset, or the Emergency Stop button is released, all axes must be "Homed" to correctly set the lathes datum points.

Instructions:

1. The system requires each of the axes to be Homed to a datum point every time power is applied to the machine. This requirement can be prompted or forced by setting Axis Parameter No.26.
2. The method and type of datum point signal to detect can be set by Axis Parameter No.27. By detecting the "Z" Pulse signal of the axis servo motor encoder after detecting the Home switch, the precision of the Home datum point is increased.
3. The direction of axis movement towards the datum point can be set by Axis Parameter No.28. D2 and D4 correspond to X & Z axes. 0 is forward, 1 is reverse.
4. The sequence of returning the X & Z axes back to the datum point can be set by Axis Parameter No.28. X axis is first when D8 is **0**, Z is first when D8 is **1**.
5. The type of switch for the datum point can be set by Axis Parameter No.29. D0 and D2 correspond to the X & Z axis. **0** means Normally Closed, **1** means Normally Open.

6. The maximum length for detecting the “Z Pulse” of a servo motor encoder can be set by Axis Parameters No.30 & No.31. The value must less than one cycle of the encoder.

7. The shifting distance after returning to a datum point can be set by Axis Parameters No.32 & No.33. After returning to the datum point, the axis moves at the Rapid rate to a coordinate value set in these parameters.

No switch for datum point:

To set the datum as a floating point, make sure to turn on the corresponding floating point function in Axis Parameter No.23, and setting Parameters No.24 & No.25, to ensure that the X axis and Z axis use a floating point as the datum point of the lathe.

To set a Floating Point datum:

1. Setting the Axis Parameter No.23 to set the axis using floating point datum. For example: To turn the X axis on is “00001000”. Z axis is “00100000”. To turn both X & Z on for FP datum, input “00101000”.)

2. Move the X axis to the desired datum position to use as the set floating point.

3. Press “Parameter”, “Axis parameter” and select parameter No.24, then “Enter”. A popup dialog box will appear with an X axis floating point coordinate. Input the value from the lathe setting coordinate.

If it is **0**, the lathe coordinate of X axis now is the datum point of X axis. The lathe returns to this position every time when homed to the datum point.

If it is **15**, the current lathe coordinate of X axis is 15.000, the distance to lathe’s datum point is 15mm.

The method to set floating point of Z axis is the same as the above to set X axis.

Operation for returning to the datum point:

In Manual mode, press  ”, and select X Y Z A axis to return to the datum point in the dialog box. Set Axis Parameter No.28 to “1” to make the Z axis return to the datum point first. In the Absolute and Relative coordinate modes, the cycle will turn green in front when the axis returns successfully to the datum point, defeat otherwise.

To abort, press “Stop” or “Reset” to stop the axis from returning to the datum point.

Attention:

Every time the system is powered up, all axes must be re-homed to their datum points, or the accuracy of the lathe processes can be degraded. If the system power is momentarily interrupted, or resulting from an accident, all axes must be re-homed to the datum point, otherwise machine damage or injury could result.

3.4.5 Handwheel

Two types: Hand Held and Panel, set by Other Parameters No.1.

Panel

Press “handwheel” and “Z” or “X” to select an axis, “  ” to adjust the gearing ratio.

Hand held

Press “handwheel” and select the axis with the Axis selector switch. Use handwheel override switch to adjust the gearing ratio.

Instructions:

The handwheel is mainly used for “Tool” setting, and the axis feed rate is related to the handwheel rotating speed. When stepper motors are used for axis control, do not rotate the handwheel too fast or positioning accuracy will be degraded.

The handwheel’s rotating speed must be lower than 200r/min. (The handwheel should have 100 pulses per rotation). Acceleration of the handwheel is controlled by Speed Parameter No.17. (larger is faster). The maximum handwheel speed is controlled by Speed Parameters No.20(X axis) & No.21(Z axis).

The handwheel has no effect during automatic machining processes. It is only enabled for working coordinates.

3.4.6 Tool

Because no two tools are the same when using multi-tool machining processes, there is a need to identify their critical dimensions prior to starting any machining. Redeeming a tool is actually moving the tool to touch the workpiece surface at a point, measuring that dimension at that point, and inputting the actual measurement of those values directly into the system. The system calculates the deviation and saves this data automatically to the corresponding tool offset register.

Press “Tool”, then choose “X” or “Z” and “Enter” to select the axis. There are two methods:

Plan A. (Recommended method)

(1) Clamp the workpiece, select appropriate spindle speed and feed rate, start spindle.
(2) Select a suitable tool to use. “Tool”, for example: T0202
(3) Using manual continuous mode, machine a small section of the cylinder or bore on the workpiece.
(4) Move the Z axis away from the part, but do not move the X axis. Stop the spindle.
(5) Measure the diameter of the workpiece. (cylinder or bore)
(6)Press “Tool”, then “X” & “Enter”, and input the above value of measurement into dialog box. Press “Enter” to confirm.
(7)Use the same method to cut the end (face) surface of the workpiece.
(8)Measure the distance from the end surface of workpiece to spindle or chuck. (Z=0)
(9)Press “Tool”, then “Z” & “Enter”, and input the measured value into the dialog box. Press “Enter” to confirm.

Tool #2 is now ready for use. (T02). Repeat the process for tools 1 to 9 as required.

Plan B

(1) Clamped workpiece, select appropriate spindle speed and feed speed, start spindle.
(2) Select the tool to "Tool", for example: T0202
(3) Using manual continuous to cut a bit of cylinder or bore on workpiece.
(4)Press "Tool", the system will appear a dialog box.
(5)X axis and Z axis both exit, stop spindle.
(6) Measure the diameter of workpiece. (cylinder or bore)
(7)Press "X" and input the above value of measurement into dialog box, press "Enter" to confirm.
(8) Use the same method to cut end surface of workpiece. Stop feeding.
(9) Press "Tool", the system will appear a dialog box.
(10) X axis and Z axis both exit, stop spindle.
(11) Measure the end surface of the workpiece to the spindle chuck (Z=0) into the distance.
(12)Press "Z" and input the above value of measurement into dialog box, press "Enter" to confirm.

The second tool is now ready for use. (T02). Repeat for Tools1 to 12 as required.
The difference between two methods:

Plan A (Recommended method)

(1) Make sure the "Tool" axis couldn't exit.
(2) The tool must touch workpiece.
(3) Use "Z" axis' direction to "Tool"

Plan B

(1) The axis can exit.
(2) The tool mustn't touch workpiece.
(3) Use "X" axis' direction to "Tool".

In the above process, the input values and lathe coordinate value's difference will be saved to the corresponding cutter compensation by the system automatically. Any inaccuracy of settings or tool wear can be modified independently for each tool. Each tool has its own coordinate system, so any tool can be modified anytime and only that tools settings are altered.

Instructions:

1. **When the same group of tools are to be used for two or more parts, processing may benefit from a complete Grid Shift of the working coordinates instead of resetting all of the tool data in the group. Confirm the value of the workpiece coordinates before setting the tools. Method of operation is as follows:**

(1) Select a tool.
(2) Press "MDI" to select the corresponding coordinate shift set. (G54 to G59)
(3) Using manual continuous, take a light cut of a cylinder or bore on the workpiece.
(4) Move Z axis away from part, (do not move X axis), and stop the spindle.

(5) Measure the diameter of the workpiece. (Cylinder or bore)
(6) Press "Setup", "X", and input the measured value. Press "Enter".
(7) Use the same method but face-cut the end surface of the workpiece.
(8) Measure to the end of the workpiece from the spindle or chuck as Z=0.
(9) Press "Setup", "Z", and input the measured value. Press "Enter".

The difference between the input values and Tool coordinate values for the new workpiece will be automatically saved to the selected corresponding parameter. Setting of the tool data must be completed before the new Coordinate Grid Shift is set. The corresponding G54 to G59 must be included as the first command line of the program.

2. **Re-setting one of the tools after a lathe crash or loss of step will not require a change to the coordinate shift.**

The method: If G53 status (Cancel Grid Shift) is active, use "Tool" instead of "Setup" in the above operations.

3.5 Auto operation:

Auto refers to the continuous automatic processing of a workpiece using the NC program. This system can begin processing from any start point, from any program line, or with any active tool. When starting from an arbitrary line or with an arbitrary tool, Absolute coordinates must initially be used in the program. Auto operation does not change the manual coordinates.

Selecting and Running a program: In the Program Interface, press "↑ ↓" to move the cursor to select the desired program. Press "C" to select the program to use in Automatic mode.

Switch coordinates display: Press "PgUp" or "PgDn" to switch between Absolute, Relative, and Machine coordinate displays.

3.5.1 Automatic process

"Single or Continuous": Press  " to switch cycle.

"Continuous": The program sequentially executes every program line to the program End or upon receiving an external instruction to stop.

"Single": The program executes one program line at a time, and waits for another operation or the "Run" button to be pressed before executing the next program line. "Coordinate or Figure": Press  " to switch cycle.

"Automatic Coordinate": The axis of coordinate will display with value.

"Automatic Figure": The axis of coordinate will display with a diagram. There are two kinds of diagrams, horizontal lathe or slant-bed lathe. The diagram display is controlled by Parameter No.3 in the Tool Parameters.

“  ”: Dry Run. The program is simulated without axis movements.

3.5.2 Processing at arbitrary program line or with arbitrary tool

A. Input the line to run

In Automatic mode, press “—” to popup a dialog box, input a Line Number, and press “Enter” to confirm. The program will run from that line.

Attention:

1/ The “Line” means the program line beginning with “N”, not the actual line in the program. The system will process to the line number you input at a speed set by parameter No.5 in the Speed parameter for (G01/G02/G03), then the program will proceed normally.

2/ To assist the user operations, the default line is the line where the program was last suspended.

3/ At the Coordinate Interface, use “N” to search for the line, and press “Reset” to return to the beginning of the program.

B. Mark the line to run from:

The system has a function to run the program from a Marked Line. In Automatic mode, press “N” to popup a dialog box, and input the marking line. Press “Enter” to confirm. Press “Run” to process the program from the line you just input. (Marked)

Attention:

1/ The “Line” means the program line beginning with “N”, not the actual line in the program. The system will process to the line number you input at a speed set by parameter No.5 in the Speed parameter for (G01/G02/G03), then the program will proceed normally.

C. Select Tool to run

The system has a function to run from a particular tool. In Automatic mode, press “G”, and the Tool Number to run. (Only input the tool number, not the associated compensation number; (For Tool 3, just input “03” and not “0304”). Press “Enter” to confirm. Press “Run” to process the program at the tool you input.

Attention: The system will process to the line number you input at a speed set by parameter No.5 in the Speed parameter for (G01, G02, G03), then the program will proceed normally.

3.5.3 Process program

Press “Auto” to change to Automatic mode and process a program. Two methods are as follows:

(1) Press the “Run” button on the controller.

(2) Activate the Run signal from a separate external controller.

3.5.4 Stop processing program

Five methods of Stopping a program are as follows:

(1) The program commands M00, M01, M02, M30, or M20.

(2) Press  " to run a current line and stop. (Single Block)

(3) Turn the Intervention switch to the middle or right.

(4) An external Switch or signal to Halt.

(5) Press "Reset" to stop all the actions of a program. (Spindle, tools and others)

3.5.5 Real-time control in automatic process

(1) Intervention switch: For suspending feeds, coordinates and stopping the spindle. There are 3 positions for this switch:

Left: Feeds, coordinates and spindle are not interrupted. Middle: Stop all feeds, but keep spindle running.

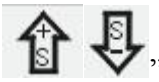
Right: Stop all feeds and spindle.

Stop only means suspend. Turn the switch to the left to continue processing. To stop the program and return to the program start, press the "Reset" button.

In Manual mode, the spindle is not affected by the Intervention switch, only by the button.

(2) "  ": Press once to increase or decrease the feed rate by 10%. When Axis Parameter No.1 is set to **0**, the range is from 0% to 150%, in 16 steps; When the No.1 axis parameter is 1, control is by an external switch. Adjust the speed of processing according to requirements for different situations.

(3) "  ": Press once to increase or decrease G00 or manual Rapid override rate by 20%. The range is from 5% to 100%, in 16 steps. Adjust the rapid override according to requirements for different situation.

(4) "  ": Press once to increase or decrease the spindle speed override by 10%. When Axis Parameter No.2 is set to 0, the range is from 5% to 150%, in 16 steps. When Axis Parameter No.1 is set to 1, control is by an external switch. Adjust the spindle speed according to requirements for different situation.

(5) Stop in the process: During continuous automatic mode, press  " to stop the process from running the next program line, and wait for operator attention.

(6) Suspend the process: Turn the intervention switch to the middle or right positions, and activate the Halt button. The processing program will stop. Press "Reset" to exit the automatic process mode and the program will reset to the first line of the process program.

(7) Continue: When the process is suspended, press "Manual" to keep feeding automatically, or to adjust the coordinates. Press "Auto" and "Run" to continue processing from the point of suspension to the end of the program.

(8) To Exit the process: Press “Reset” during processing, suspension or feeding.

3.5.6 Operating in MDI mode

In the manual or automatic coordinate conditions, press “M” to change to MDI mode. A line of code can be input in the MDI mode, and processed by pressing the “Run” button. To exit without processing, press the “Esc” button.

3.5.7 Handwheel operation mode

Press “Handwheel” in automatic mode. The program of turn handwheel is processing automatically, the speed is related to the speed of “F”, feed override and turn handwheel fast or slow. This mode is for trying to process in running program usually.

Attention: The acceleration, deceleration and maximum speed of the running handwheel are controlled by Speed Parameters No.17 No.18 No.19 No.20 No.21 & No.43, and Processing Parameter No.23. The acquiescent acceleration, deceleration, and the speed of G00 when the parameter is set to be invalid.

3.5.8 The function of DNC

This system has a User Storage space of 32 Mbits. Use the DNC facility when the available program space is exceeded, or a single program is too large to fit into this space. Switch to RS232 or USB and use the DNC function of this system.

A. Instruction for using RS232-DNC:

1. Use the dedicated communication line to connect the computer to the system, and set the corresponding communication interface and speed to suit.
2. Use the dedicated communication software for this system by setting the computer and corresponding communication interface speed. Press “Send CNC program file”, select the program file to process, and enter the status of the program file to send.
3. Enter the program file interface of the NC system, and press "L" to enter the status of the linked process. “RS232-DNC” is now displayed in the upper right corner of the display interface. Press “Run” to use the linked process in Automatic mode.
4. Turn the “Intervention switch” to the middle or right to stop running the process from the linked system. Press “Stop” or “Reset” to exit the linked process.

Attention: 1. The NC system and PC baud rates and settings must be the same when using the serial port to send files.

2. The communication cable must not be more than 10 meters long.

3. Only the dedicated communication software of this system can send programs to a user’s computer. Set the sending speed of the PC and the NC equally, or the file transfer will fail.

B. Instruction for using USB-DNC:

USB-DNC is available via U-disk transfer. Insert the U-disk into the system port, switch on the system, and select the program to execute from the U-disk.

Press “B” to open U-disk in Program Interface, select the desired program, and press “C” to execute this program. Press “Auto” to get into Automatic mode and press “Run” to process the program.

Attention: 1. Do not unplug the U-disk during the USB-DNC process, or the process will fail.

2. After selecting the program, press “P” to compile the USB-DNC program and check for errors before running.

3. Return to the system Program Interface from the U-disk Interface after the USB-DNC process finishes.

3.6 Operate safety, prompt alarm

3.7 Emergency stop

Press “ ” if an Emergency condition occurs. The system will stop all functions and display “Emergency stop” on the interface. This condition is maintained until the button is reset. M67 inputs an effective signal if Other Parameter No.29 is active.

If the “ ” button is pressed during the automatic lathe process, the system coordinates and lathe’s position may change. To ensure the system and coordinates are correct before continuing the processing, re-home all axes to their datum points and re-start the program.

An external Emergency Stop button can be controlled by Other Parameters No.27, and set as Normally Open, or Normally Closed. (For safety reasons, Normally Closed is standard.)

3.6.2 Reset system

Press “Reset” to stop the current operation at any time while the system is running. This will stop all the actions of the lathe (spindle, tools and other functions) in Automatic or Manual modes, but the coordinates will be retained, so there is no need to re-home the axes to the datum point.

3.6.3 Alarm

The screen displays any error information and flashes when the lathe detects an alarm, the program will stop running, and the coordinate displays stop. Check the

reason for alarm and rectify the problem to continue. The M67 signal is effective when Other Parameters No.29 is set to "**1**".

(1) X and Z axis are limited forcedly positive: An X or Z axis Positive Limit Switch has been activated.

(2) X and Z axis are limited forcedly negative: An X or Z axis Negative Limit Switch has been activated.

(3) Spindle and inverter (frequency changer) alarm: An alarm has occurred in the Spindle Controller. (ALM1)

(4) No.0 alarm: An alarm has occurred on the lathe spindle. (ALM2)

(5) X and Z axis' driver alarm: An alarm signal has been received from the Axis Servo Drivers. (ALM). Press "B" in diagnosis mode to input an INTH signal and reset the servo drivers.

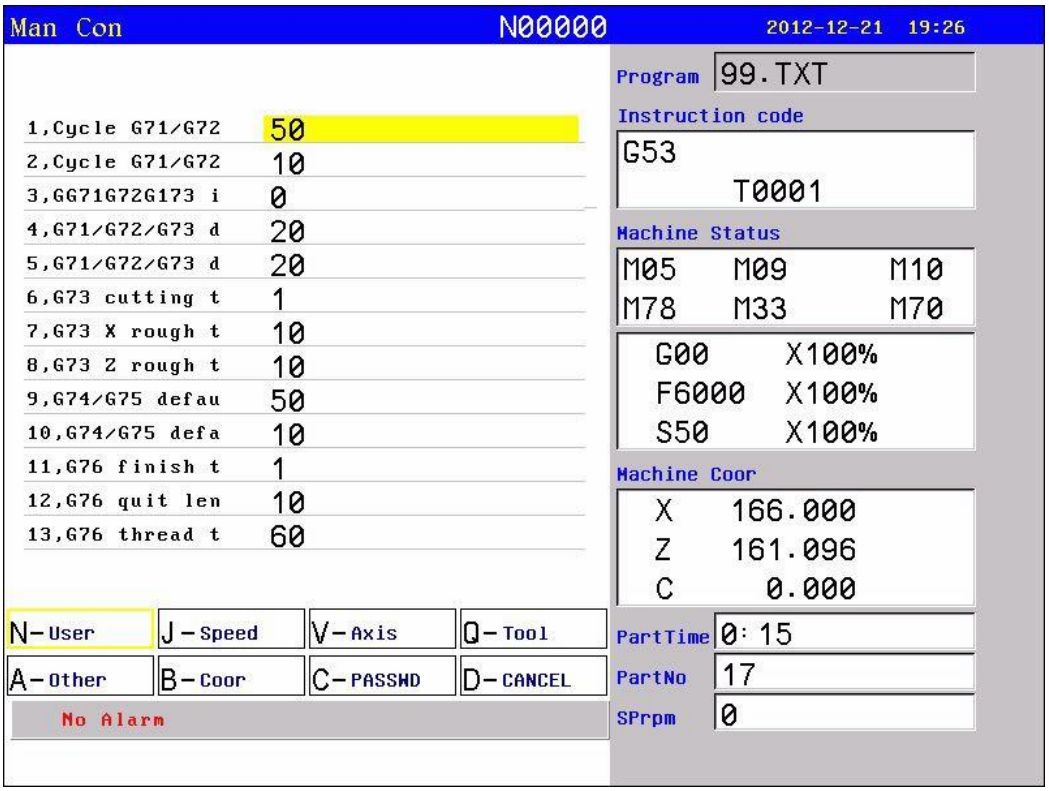
(6) No.5 alarm for door switch: An M12 (door switch) alarm is effective.

(7) +5V is under voltage: System Supply +5v rail is low.

(8) Emergency stop: An Emergency Stop Button has been pressed.

3.7 Parameter operation

Press "parameter" to enter the Parameters status and change any of the system Parameters from any status condition. The seven (7) types of Parameter in this system are "Processing parameters" "Speed parameters" "Axis parameters" "Tool parameters" "Other parameters" "Coordinates" and "Password".



Press N J V Q A B C to select the desired interface from the Parameter Interface. Press "↑ ↓" to select the parameter number, and press "Enter" to popup a dialog box for data input. Press "Enter" again to set the parameter.

Details of parameters are as follows: See appendix for detailed explanations

3.7.1 User parameters (processing)

NB. Some parameters have multiple entries like D3X;D5Z – these are binary format and are in reverse order of D7,D6,D5,D4,D3,D2,D1,D0 – note the D0!

1, Cycle G71/G72 default feed thickness (10um) [X axis radius]

Cycle G71 X axis feed thickness;

Cycle G72 Z axis feed thickness;

2, Cycle G71/G72 default backward distance. (10um) [X axis radius]

Cycle G71 X axis backward distance;

Cycle G72 Z axis backward distance.

3, G71 G72 G173 instruction [**1** means Yes, **0** means No] "1" means G71/G72/G73 will finish the machining.

4, G71/G72/G73 default X remain (10um)

5, G71/G72/G73 default Z remain (10um) 6, G73 Number of passes

7, G73 X rough thickness (10um) [X axis radius] 8, G73 Z rough thickness (10um)

9, G74/G75 default feed thickness (10um) [X axis radius]

10, G74/G75 default backward distance (10um) [X axis radius] 11, G76 Number of finish passes

12, G76 Exit length (1/10 lead)

13, G76 Thread tooth angle (degree) [0～180°]

14, G76 Minimum cutting depth (10um) [X axis radius]

15, G76 Finish turn remaining (10um)

16, X programming mode [**1** means Radius, **0** means Diameter]

17, Active program needs Spindle running [**1** means Yes, **0** means No]

17-1, Active program needs Spindle M10 [**1** means Yes, **0** means No]

18, Set M20 the time of auto-running [Negative means infinite loop]

20, G92 Exit length (1/10 lead)

21, G01/G02/G03 line delay (ms) [>100]

22, G00 line delay (ms) [>100]

23, Handwheel acceleration [50-100]

24, G76 Q [**8** means thick forward number]

200, System screen protect times [>=2minutes]

201, G92/G76 delay time (ms) [>100]

202, System inner parameter

203, Enable pause

210, Enable Graphics Display Area [**0**=Auto **8**=Manual]
211, X axis negative end Factory set to 1 ?
212, X axis positive end Factory set to 1 ?
213, Y axis positive end Probably should read negative end Factory set to 1 ?
214, Y axis positive end Factory set to 1 ?
215, Z axis positive end Probably should read negative end Factory set to 1 ?
216, Z axis positive end Factory set to 1 ?
230, Select executive program through input point ?
231, "Delete" mode cursor position [0=Cursor to right 1=Cursor to left]
232, Check spindle running before tapping Factory set to 1 ?

3.7.2 Speed parameter

1, X-axis's G00 Rapid speed (mm/min)
2, Z-axis's G00 Rapid speed (mm/min)
3, Manual Maximum feed rate (mm/min)
4, Auto Maximum feed rate (mm/min)
5, G01/G02/G03 default speed (mm/min)
6, Null run speed (mm/min)
7, Feed axis manual speed (mm/min)
8, Spindle manual speed (rpm)
11, Limit G1, G2, G3 axis speed [**1** means Yes, **0** means No]
12, X G1, G2, G3 max feed rate (mm/min)
13, Z G1, G2, G3 max feed rate (mm/min)
14, X acceleration [1～99999]
15, Z acceleration [1～99999]
16, Auto run acceleration [1-500]
17, Handwheel acceleration [500 - 30000]
18, Run program Handwheel acceleration [>500]
19, Run program Handwheel G00 speed (mm/min) [>10]
20, Handwheel X limit speed (mm/min)
21, Handwheel Z limit speed (mm/min)
22, Make thread Z acceleration [Servo motor is 0]
23, Make thread X acceleration [Servo motor is 0]
24, Servo motor screw thread X axis Back speed
25, Stepper motor screw thread X axis Back rise speed
26, Stepper motor screw thread X axis Back Max speed
27, Acceleration type [**0** means line, **8** means curve]
28, Curve initial acceleration [>=10]
29, Curve acceleration [>=10]

30, Curve max acceleration [>=500]
31, X Home rapid speed (mm/min)
32, X Home reverse speed (mm/min)
33, Z Home rapid speed (mm/min)
34, Z Home reverse speed (mm/min)
35, G96 spindle min speed (rpm)
36, Spindle first gear max speed (rpm)
37, Spindle second gear max speed (rpm)
38, Spindle third gear max speed (rpm)
39, Spindle fourth gear max speed (rpm)
40, Secondary Spindle max speed (rpm)
41, G02/G03 reverse compensation mode. (**0** means A; **8** means B)
42, Mode B reverse compensation speed (mm/min)
42-1, Mode B reverse compensation initial feed speed (mm/min)
42-2, Mode B reverse compensation acceleration (mm/min/s)
43, Handwheel stop speed (mm/min) [>100]
49, Speed Mode [1=Yes 0=No]
50, Handwheel stop speed mm/min [>100]
51, Follow the tapping cutter when spindle speed (rpm) [>1]
52, When tapping, backlash compensation (pulse)
53, Follow the tapping cutter withdrawal before reversing (um) [10-5000]
54, Tapping back speed (mm/min) [>=60]
58, Forced limit drop speed critical (mm/min)
60, G01/G02/G03 smooth running [1=No 60=Yes]
61, G01/G02/G03 running time normal [2-50]
101, X-Beginning feed speed (mm/min) [>1]
102, Y-Beginning feed speed (mm/min) [>1]
103, Z-Beginning feed speed (mm/min) [>1]
111, X-Jump speed at continuous track (mm/min) [>1]
112, Y-Jump speed at continuous track (mm/min) [>1]
113, Z-Jump speed at continuous track (mm/min) [>1]
200, G00 continuous run is valid [1=N 16=Yes]
210, Thread is waiting for spindle speed [**0**=No **1**=Yes]

3.7.3Axis parameter

1, Feed axis band switch [**1** means Yes, **0** means No]
2, Spindle band switch [**1** means Yes, **0** means No]
3, X-axis`s negative scope (mm)
4, X-axis`s positive scope (mm)

5, Z-axis`s negative scope (mm)
6, Z-axis`s positive scope (mm)
7, Spindle stop time (10ms)
8, Spindle stop long signal [**0** means No, **1** means Yes]
9, Check SP encoder [**0**=No **1**=Yes]
10 SP encoder pulses
11, Soft limit invalid [D2X; D3Y; D4Z; D5A; **0** valid, **1** invalid]
12, X-axis`s reverse compensation (um)
13, Z-axis`s reverse compensation (um)
14, X-axis's direction signal [**1** means normal, **0** means reverse]
15, Z-axis's direction signal [**1** means normal, **0** means reverse]
16, Close feed electronic gear [**1** means Yes, **0** means No]
17, X-axis electronic gearing numerator (1-999999)
18, X-axis electronic gearing denominator (1-999999)
19, Z-axis electronic gearing numerator (1-999999)
20, Z-axis electronic gearing denominator (1-999999)
21, XZ positive limit [**0** open, **1** close]
22, XZ negative limit [**0** open, **1** close]
23, Float zero bit parameter [D3X; D4Y; D5Z; D6A; **0** machine Zero; **1** floating Zero]
24, X coordinate floating Zero set value
25, Z coordinate floating Zero set value
26, Feed axis home [**1** =Not used, **0** =Prompt, **8** =Compulsory, **9** =Must comply]
27, Feed axis home mode [**0** =reverse & check, **1** =No reverse but No check, **2** No reverse check, **3** No reverse and No check]
28, Home axis in reverse direction [D2X; D3Y; D4Z; D5A; **0** Positive; **1** Negative]
29, Home switch set [D0X; D1Y; D2Z; D3A; **1 =**Closed; **0** =Open]
30, X check zero max length (100um)
31, Z check zero max length (100um)
32, X Home offset (10um)
33, Z Home offset (10um)
50, Have Spindle class control [**1** means open, **0** means closed]
51, Spindle class speed (1/100rpm)
52, Spindle class direction [**0** means M03, **1** means M04]
53, Spindle class stop time (10ms)
54, Spindle class time (10ms)
55, Spindle stop time (10ms)
56, Spindle manual point [**8**=M04]
68, XYZA Reverse delay time (ms)
80, X Z Axis coordinate plan [D2 =Z workpiece, D3 =X workpiece, D4 =Z Tool, D5 =X Tool, D6 =Z (C) Rotary, D7 =X (A) Rotary]
109, C(Y) G1, G2, G3 max speed (mm/min)

111, Handwheel C/Y limit speed (mm/min)
117, C (Y) – axis negative scope (mm)
118, C (Y) – axis positive scope (mm)
119, C (Y) coordinate float zero set
200, System inner parameter [**1**]
404, Spindle motor direction (**0** reverse, **1** normal)
405, Spindle electronic gearing (**0** Yes, **1** No)
406, Spindle electronic low gear numerator (1-999999)
407, Spindle electronic low gear denominator (1-999999)
408, Spindle electronic high gear numerator (1-999999)
409, Spindle electronic high gear denominator (1-999999)
410, Interpolation tap SP name [91 X, 92 Y/C, 93 Z, 94 A, 95 B]
411, Interpolation tap mode [**0** follow encoder; **4** interpolation to Spindle]
412, Spindle gear teeth (<P413)
413, Encoder gear teeth (>P412)
414, A-Axis is moving by [**7** by X, **8** by Y, **9** by Z]

3.7.4Tool parameter

1, Active tool function [**1** means Yes, **0** means No]
2, Active tool number
3, Lathe type
4, Tool positive rotation time max (s)
5, Delay time after positive tool rotation (ms)
6, Delay time after tool rotation stops (ms)
7, Clamping time after reverse tool rotation (ms)
9, TOK (Tool Clamped) signal expected (**1** means Yes)
10, C Tool radius compensation established (**0** means A, **1** means B)
11, C Tool radius compensation cancel (**0** means A, **1** means B)
20, Active tool mode [**1** means normal, **0** means coding tool]

3.7.5 Other parameter

1, Set sub-panel type [**1** means Hand-Held, **0** means Panel]
2, Lathe chuck clamping direction [**1** external, **0** internal]
3, Use control switch [**1** =Yes, **0** =No]
4, Auto lubricator fitted (**0** =Yes, **1** =No)
5, Auto lubricator Run Time (0.01s)
6, Auto lubricator stop time (s)

7, Door switch checking M12 (**0** =No, **1** =Yes)

8, Door switch type (**0** = Normally Open, **1** = Normally Closed)

9, **Bit parameters**

D0：Null;

D1：“**1**” Start CNC system. Clear part Number;

D2：“**1**” Insert space before Alpha characters when editing program;

D3：Null;

D4：Null;

D5：“**1**” Do not stopping Spindle or coolant when “Reset” is pressed;

D6：“**1**” G00 X and Z speed by oneself; ????

D7：“**1**” Tool redeem by oneself; ????

D8：“**1**” Save (M10/M11) Spindle chuck state when power off;

D9：Tool redeem input Mode-1 or Mode-2;

D10：“**1**” Program edit automatic compositor Line;

D11：“**1**” First SP +10V output from second output port;

D12：“**1**” Shield skip function (“/” is ignored in program lines);

D13：“**1**” Shield “Home” function;

D14：“**1**” Shield “Run” key;

D15：“**1**” Tool redeem display Relative, “**0**” = Absolute;

10, Auto parts counter increment [**1** means Yes, **0** means No]

11, Program editing. Increment line numbers by

13, Does lock for spindle & chuck M10

13-1, Does lock for spindle M79

14, Keys for lubrication & coolant active while running (**0** means No)

15, Chuck clamping. M10 - M11 Clamp/Unclamp check (**1** means Yes)

16, Tailstock extend M79, retract M78 check (**1** means Yes)

17, Servo ALM1 signal (**0** = Normally Open, **1** = Normally closed)

18, Spindle ALM2 (**0** = Normally Open, **1** = Normally closed)

19, Tool ALM3 (**0** = Normally Open, **1** = Normally closed)

20, Chuck control signal M10/M71 (**0** single, **1** double)

21, Tailstock control signal (0 single,1 doubleM79/M73)

22, External chuck control (**0** =No, **1** =Yes)

23, External tailstock control (**0** =No, **1** =Yes)

24, M10-M11 short signal time (s)

24-1, M10 long signal

24-2, M71 long signal

24-3, Chuck M10 or M11 at boot

25, M79-M78 short signal time (s)

25-1, M79 long signal

25-2, M73 long signal
26, Emergency Stop (**0** = Normally Open, **1** = Normally closed)
27, Emergency Stop-2 (**0** = Normally Open, **1** = Normally closed)
28, Run status output M69, STOP output M65 (**0** Invalid, **1** Valid)
29, Alarm status output M67 (**0** Invalid, **1** Valid)
30, Set User Language (**1** means Chinese, **0** means English)
31, Enable I/O from PLC program
32, Enable High speed I/O PLC program
33, HY make Run signal [**1** means Yes, **0** means No]
34, HA make Stop signal [**1** means Yes, **0** means No]
35, Soft-limit without home in Manual [**1** =Yes, **0** =No]
36, Set system time [Year-Month-Day-Hour-Minute]
37, Baud Rate of RS232 [**0**=7200；**1**=9600；**2**=14400；**3**=19200；**4**=38400；**5**=57600；**6**=115200]
38, Lock Manual Tailstock function key [**8** =Yes]
39, Special parameter
40, Special parameter
41, Bake current parameters
42, Resume original parameters
50, Run from mid program ask for going to last line point
120, Manual axis moving keys reverse
200, X axis encoder feed-back error limit. (pulses)[>1]
202, Z axis encoder feed-back error limit. (pulses)[>1]
205, X axis encoder feed-back Stop Alarm limit. (pulses)[>1]
207, Z axis encoder feed-back Stop Alarm limit. (pulses)[>1]
210, X axis feed-back electronic gear numerator
[Automatically calculate L screw pitch (um) to M encoder pulses.]
212, Z axis feed-back electrical gear numerator
215, X axis feed-back electrical gear denominator
217, Z axis feed-back electrical gear denominator
300, Feed axis fitted with Absolute Encoder motor or not.
[X=D2, Y=D3, Z=D4, A=D5, "**0**" means not match, "**1**" means match.]
500, Machine tool number
501, Interface Display [**1**=White **8**=Black]
601, Define as Stepper
602, Define as Servo
900, User-defined dialogue box [**1**=No **4**=Yes]
901, Axis home order
910, M18/M22/M24 high speed input [**1**=Yes]
911, Enable M18 for teaching, M28 for recording function [**0**=No **1**=Yes]
912, "Reset" to reset output interface [**1**=Yes]

3.7.6 Coordinate

1, X of work coordinates G54 (mm) [Modify increment, input “E” to clear]
2, C(Y) of work coordinates G54 (mm) [Modify increment, input “E” to clear]
3, Z of work coordinates G54 (mm) [Modify increment, input “E” to clear]
4, A of work coordinates G54 (mm) [Modify increment, input “E” to clear]
6, X of work coordinates G55 (mm) [Modify increment, input “E” to clear]
7, C(Y) of work coordinates G55 (mm) [Modify increment, input “E” to clear]
8, Z of work coordinates G55 (mm) [Modify increment, input “E” to clear]
9, A of work coordinates G55 (mm) [Modify increment, input “E” to clear]
11, X of work coordinates G56 (mm) [Modify increment, input “E” to clear]
12, C(Y) of work coordinates G56 (mm) [Modify increment, input “E” to clear]
13, Z of work coordinates G56 (mm) [Modify increment, input “E” to clear]
14, A of work coordinates G56 (mm) [Modify increment, input “E” to clear]
16, X of work coordinates G57 (mm) [Modify increment, input “E” to clear]
17, C(Y) of work coordinates G57 (mm) [Modify increment, input “E” to clear]
18, Z of work coordinates G57 (mm) [Modify increment, input “E” to clear]
19, A of work coordinates G57 (mm) [Modify increment, input “E” to clear]
21, X of work coordinates G58 (mm) [Modify increment, input “E” to clear]
22, C(Y) of work coordinates G58 (mm) [Modify increment, input “E” to clear]
23, Z of work coordinates G58 (mm) [Modify increment, input “E” to clear]
24, A of work coordinates G58 (mm) [Modify increment, input “E” to clear]
26, X of work coordinates G59 (mm) [Modify increment, input “E” to clear]
27, C(Y) of work coordinates G59 (mm) [Modify increment, input “E” to clear]
28, Z of work coordinates G59 (mm) [Modify increment, input “E” to clear]
29, A of work coordinates G59 (mm) [Modify increment, input “E” to clear]

3.7.7 Password

Password setting includes:
1, Enable CNC Companies password ?
2, Enable Machine Companies password ? Original password is “NEWNEW”.
3, Enable User’s password ? Original password is “KERKER”.
4, Modify CNC Co.’s password:
5, Modify Machine Co.’s password:
6, Modify User’s password:
7, Time in Operation: (days)
8. Version of Software

3.8 Set parameters of tool redeem

Press “Redeem” to enter Redeem interface from any interface. The seven (7) options are “Radius compensation” “Redeem” “Clear all values” “Clear current value” “Measure tool” “Posit tool” and “Set”, called by pressing the corresponding “N” “J” “V” “Q” “A” “B” or “C” buttons. Press “Esc” to return to the primary menu interface.

3.8.1 Radius compensation

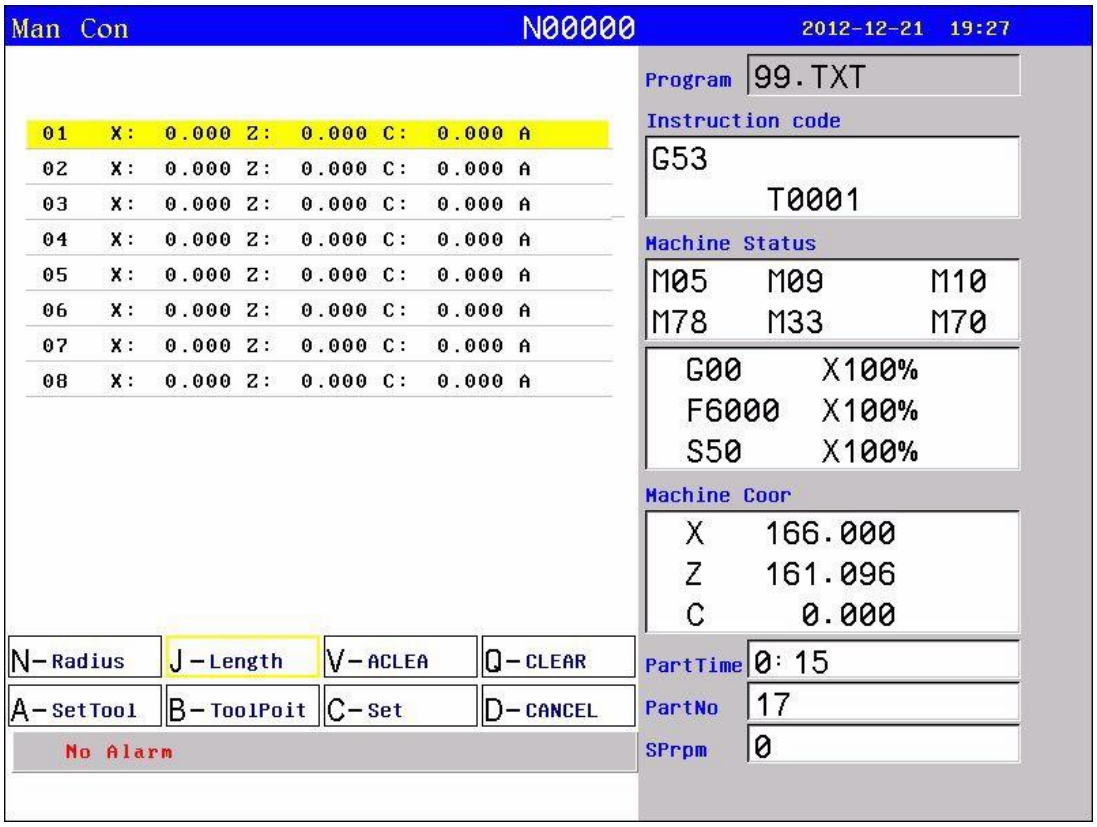
Press “N” to enter the Radius Compensation interface from the Redeem interface.

This parameter is used to define the corner radius of the tool.

Setting method: Press “↑ ↓” to move the cursor to the desired tool, and press “Enter” to popup a dialog box. Input the corresponding tool radius, and press “Enter”.

Press “V” or “Q” to clear all initial or current tool data to 0.

Attention: Redeem number couldn’t correspond to tool, every tool could use any redeem number, radius compensation correspond to redeem number, so the number of redeem is the same as the number of radius compensation. ????



3.8.2 Length of redeem

Press “J” to enter a Length in the Redeem Interface. The parameter is used to set or modify the tool data for new or reset tooling.

Method of modifying the tool length:

Press “↑ ↓” to move the cursor to the corresponding tool number, and press “Enter” to popup a dialog box. Input the required axis into the dialog box and input the modified value, (example [Input 0.05 to add 0.05, input -0.05 to subtract 0.05]), then press “Enter” to confirm. The system calculates the new redeem values after setting.

Method of resetting the length:

Move the axis to a position suitable for measuring the corresponding tool coordinate. Press "↑ ↓" to move the cursor to the corresponding tool number and press "A" to popup a dialog box. Input the axis to reset into dialog box, and input the measured value of the workpiece in the corresponding axis. Press "Enter" to confirm. The length compensation for the corresponding axis has now been reset. The system automatic refreshes the current redeeming value after setting is finished.

Method of initializing the length compensation value of a tool:

Press "V" or "Q" to initialize all of the length compensations or current length compensation.

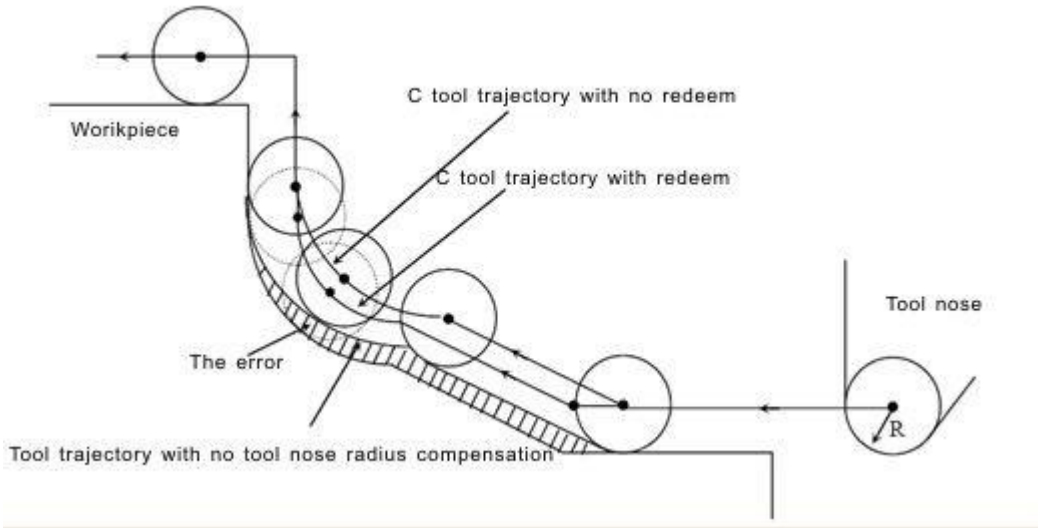
3.8.3 Posit tool

Press "B" to enter the Posit Tool interface from the Redeem Interface. These parameters are used to set the kind of tool when adopting radius compensation of tool. Method of setting: Press "↑ ↓" to move the cursor to the corresponding tool number, and press "Enter" to popup a dialog box. Input the code for the corresponding tool type, and press "Enter" to confirm.

Press "N" to initialize all the tool types to 0.

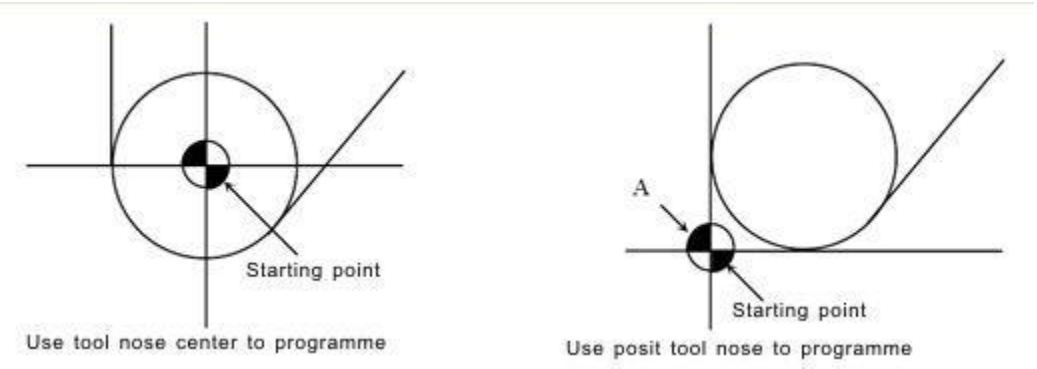
Introduction of posit tool

Programming a part to the correct shape without compensating for the Tool Nose Radius offset is very difficult. The Tool Compensation function calculates the correct tool path automatically.



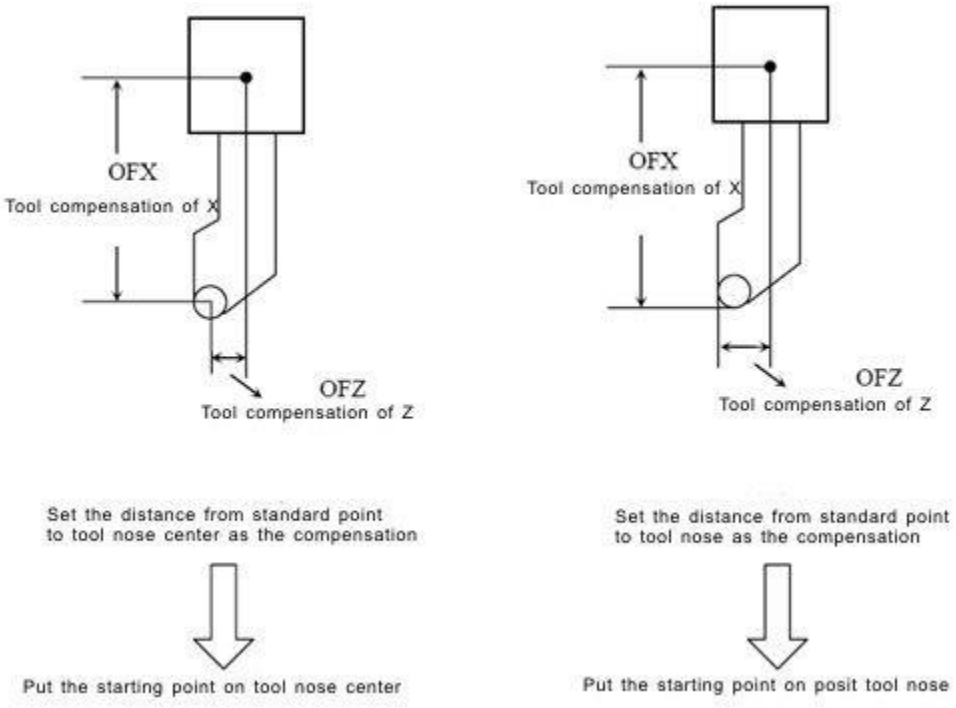
A. Posit tool

The nose of tool point (A) is non-existent as shown in the follow picture. Setting Posit Tool is much easier than setting from the actual center of tool. By using Posit Tool to program a part, we do not need to consider the tool radius. The tools shown in the diagram below indicate the different origins and tool position relationships.

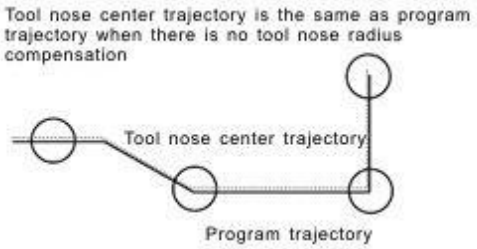


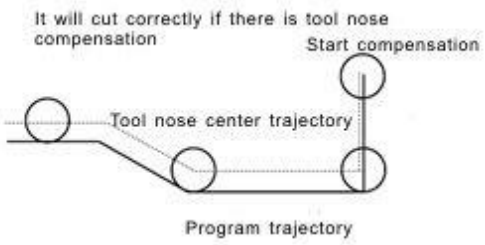
Attention: For some lathes with an electrical datum point, a standard point (like center nose of a tool) can be the starting point. The distance from the standard point to the radius center of tool rest or posit tool can be the offset of the tool.

When the tool rest is set as a beginning point.

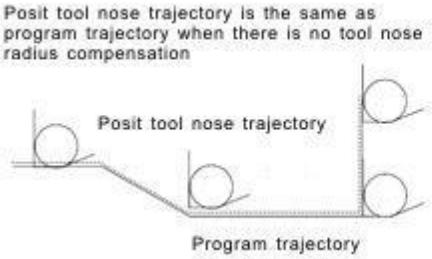


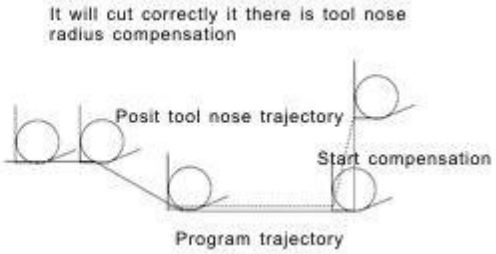
Programming from the centre of a tool nose:





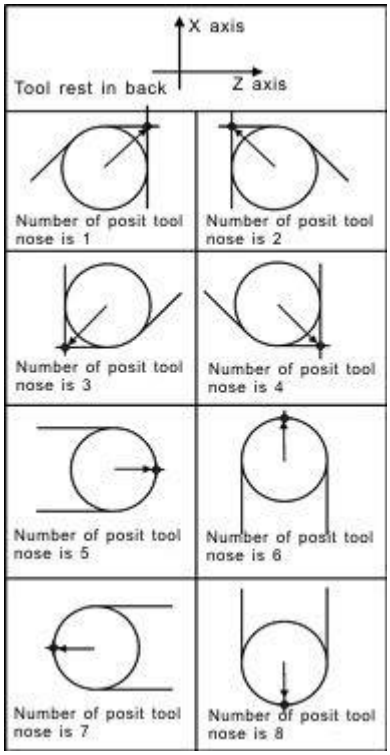
Using the posit tool to program



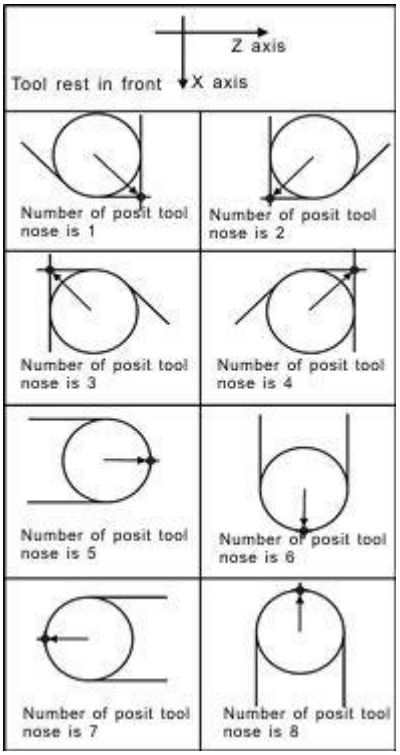


B. Direction of posit tool

From the central nose of tool to see the direction of posit tool is determined by the direction of cutting tool, so must set before as the same time with the value of compensation. The direction of posit tool can be selected by the 8 kinds of corresponding number as shown below.



The direction of posit tool for a backward tool rest



The direction of posit tool for a forward tool rest

Using No.7 posit tool or No.9 when the central nose of tool is coinciding with starting position. Set the number of posit tool into tool parameter in every number of offset.

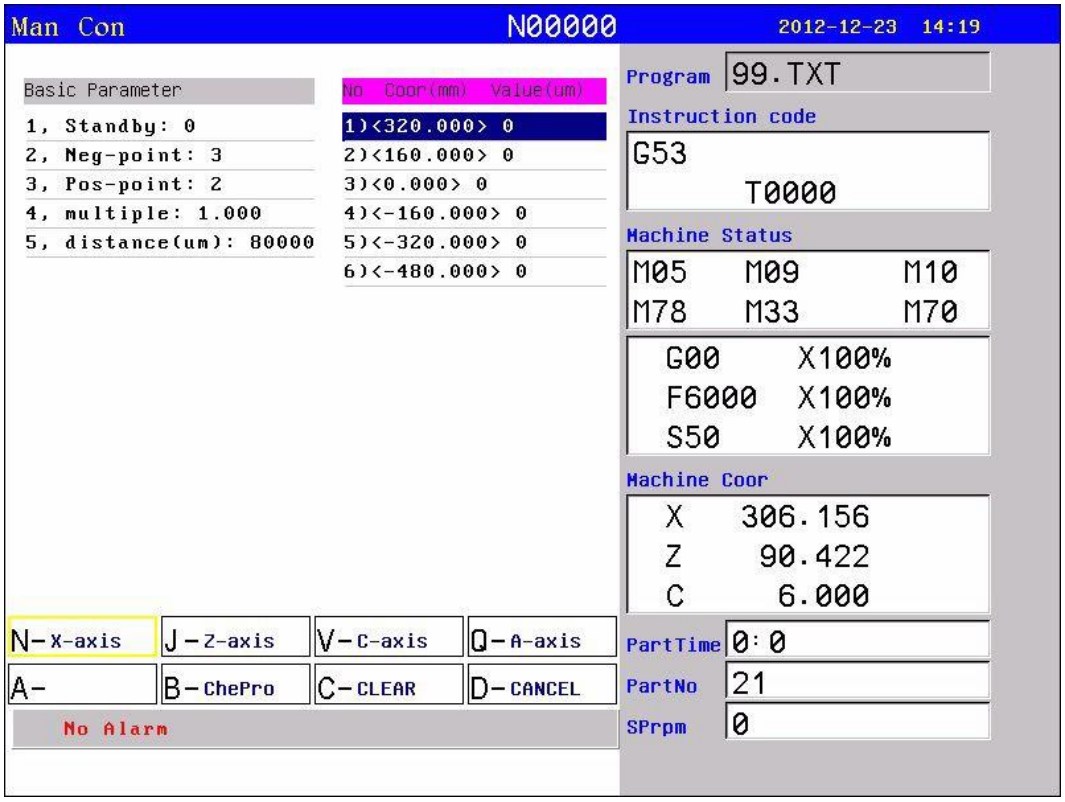
3.8.4 Set quantity

Press “C” to popup a dialog box in the Redeem Interface to set and manage the total number of tools.

Include the sum of tools in an electrical tool rest and a tool-post.

3.9 Screw compensation

Press “Parameter” twice in Parameter Interface to enter the Screw Compensation Interface and set the screw compensation.
Screw compensation is used for automatically compensating for any errors in the screw pitch, which reduces the influence of screw pitch errors on the lathe accuracy. The system adopts a storage mode of screw compensation: The lathes datum point is used as the starting point when debugging. Measured the error curve of the screw, define the correctional curve according to the error curve, input the values of correctional curve into the correctional parameters, and the system will automatically compensate for the screw pitch errors during operating modes.



Screw compensation interface:

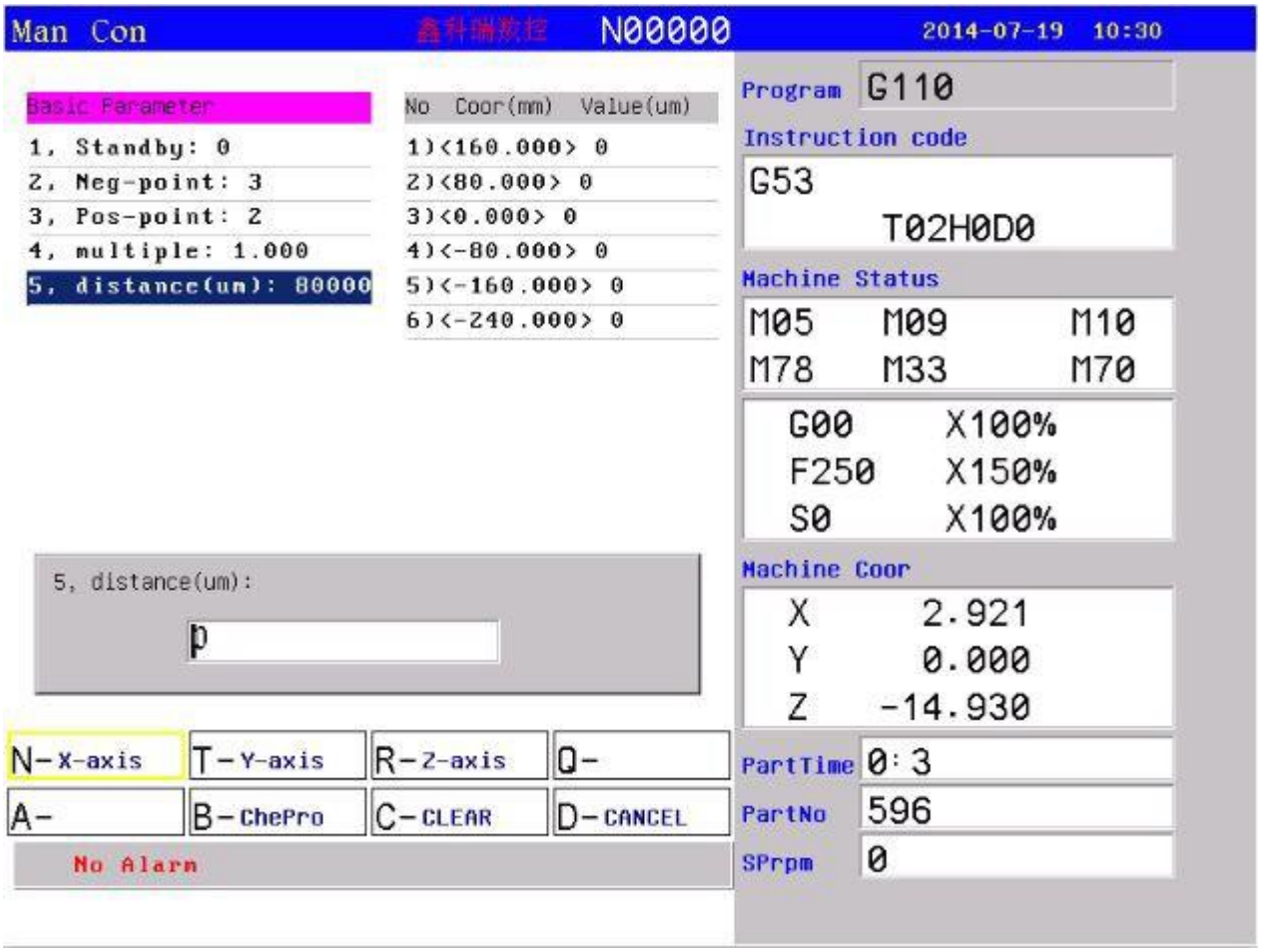
Screw compensation is set for each axis and stored as a unit. Set the X & Z axis separately, by pressing “N” or “J” to switch. Each axis of the Screw Compensation Interface has two areas (basic parameter and the compensation setting). Press “→ ←” to move the cursor between items.

Basic parameter:

Press "↑ ↓" to select the current Basic Parameters. Press "Enter" to pop-up a dialog box to input the error compensation of every axis and input the basic information of screw compensation.

Set compensation:

In the Compensation Setting area, it displays the compensation value for each axis error compensation point of the screw pitch. Press "↑ ↓ PgDn PgUp" to select a current compensation point, and press "Enter" to popup a dialog box. Select the compensation point, and input the new compensation value.

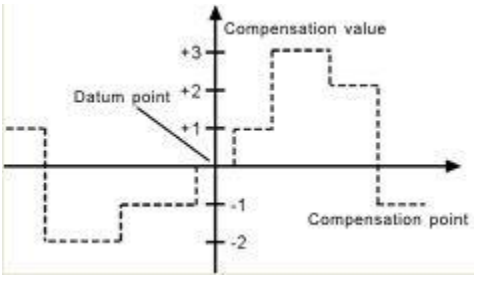


Test program Automatic Generation

Automatic generation of a program for a laser interferometer to check ball screw compensation. Enter the Screw Pitch Interface and set a basic parameter, press Checking Program to popup a dialog box and press "Enter" to generate a corresponding checking program for the screw compensation.

The number of compensation points can be set freely from a maximum number of 300 points per axis. The basic parameter of every axis' error compensation of screw pitch includes as follows:

1. Reserve.
2. Backward checking points.
3. Forward checking points.
4. Compensation override.
5. The spacing of compensation points (um).



The system automatically calculates the error compensation points for each axis at each position of the screw pitch according to basic parameters. Every axis's error compensation point spacing is uniform, so users can input a compensation value for each point. (This system requires the input of absolute values, relative to the datum point).

The compensation points are uniformly spaced for each axis.

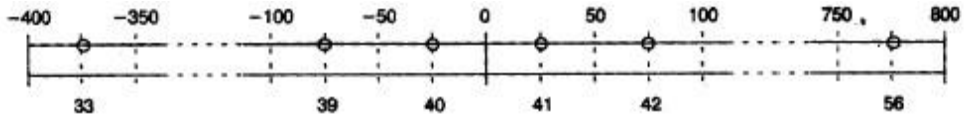
For example:

Example 1: Linear axis: when length of travel is -400mm～+800mm, the point spacing is 50mm, and reference point compensation is No.40. The system calculates that the Compensation point at the farthest end in the negative direction is:

Machine negative travel/point interval +1=40-400/50+1=33.

Compensation point of farthest end in positive direction is: Machine positive travel/point interval +1=40+800/50=56.

The corresponding relationship between machine coordinate and compensation point is:



Output compensation for value in 0 position parameters is set as follows：

Compensation point No. of reference point: 40

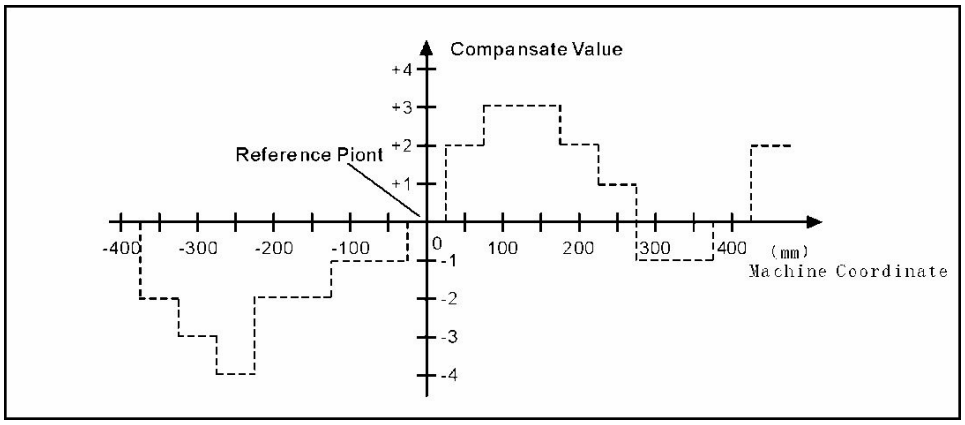
Compensation point No. of farthest end in negative direction: 30 Compensation point No. of farthest end in positive direction: 56 Compensation override: 1

Compensation point interval: 50000 Corresponding compensation point and value:

The compensation value in corresponding compensation point.

No.	33	34	35	36	37	38	39	40	41	42	43	44	45	46	47	48	49		56
Value	+2	+1	+1	-2	0	-1	0	-1	+2	+1	0	-1	-1	-2	0	+1	+2		+1

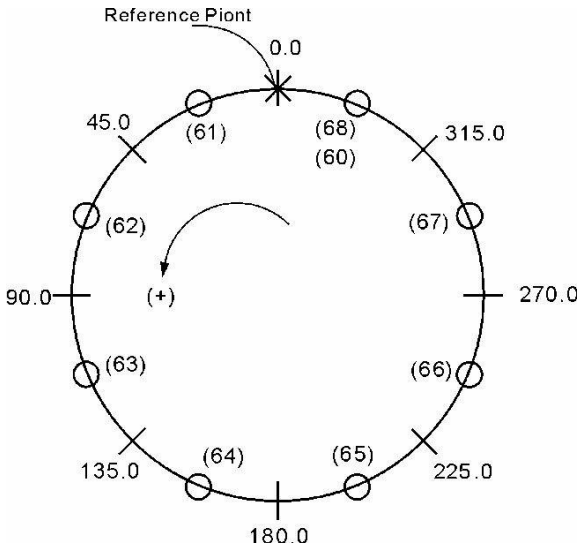
The contrasted chart of compensation points and value as follows:



Example 2: Rotary axis, where the movement per revolution is 360°, and the points interval is 45°. The reference point compensation is No.60, and the Compensation point No. at farthest end in the negative direction is usually the same as the Reference Point Compensation Point No.

Compensation point No. at the farthest end in the positive direction is:

Reference compensation point No.+ movement per revolution/compensation point interval=60+360/45=68.



Corresponding Machine coordinate and compensation point No. is:

Note: Input values for a small circle. If the total amount from 61 to 68 doesn't equal 0, the accumulated pitch error per revolution will deviate, so input the same values in 60 and 68.

Parameter sets as follows:

Compensation point No. of reference point: 60

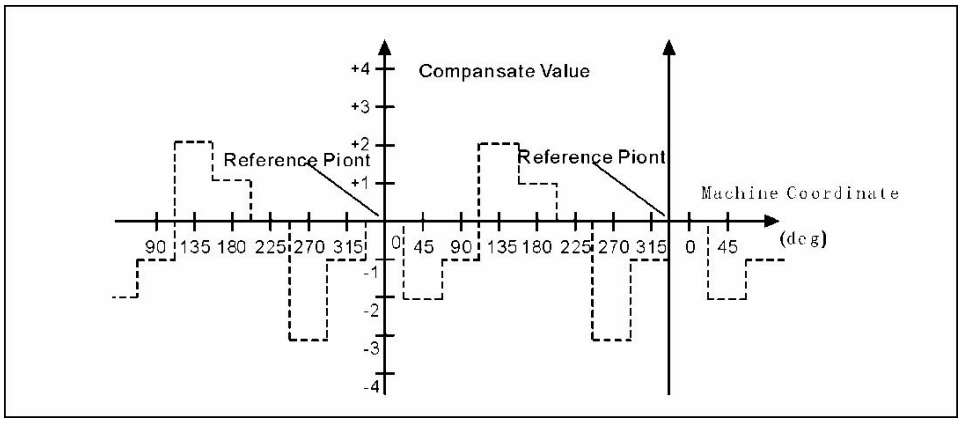
Compensation point No. of farthest end in negative direction: 60 Compensation point No. of farthest end in positive direction: 68 Compensation percentage: 1

Compensation point interval: 45000

Output compensation value at corresponding point:

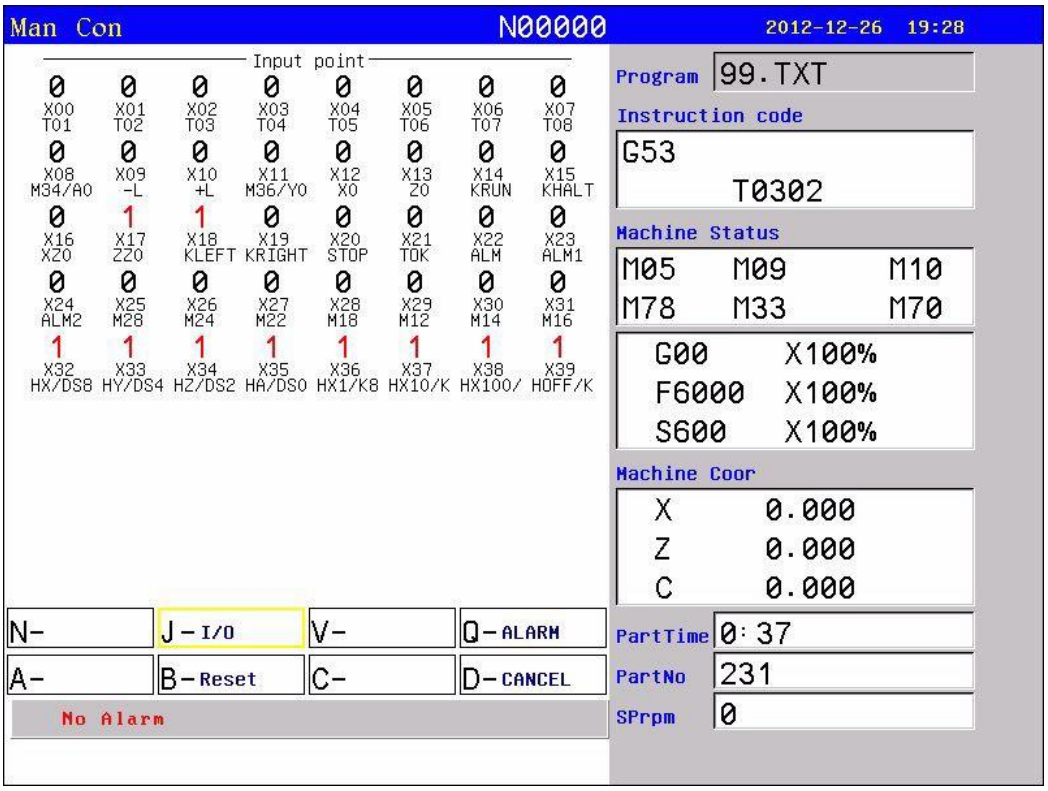
NO.	60	61	62	63	64	65	66	67	68
VALUE	+1	-2	+1	+3	-1	-1	-3	+2	+1

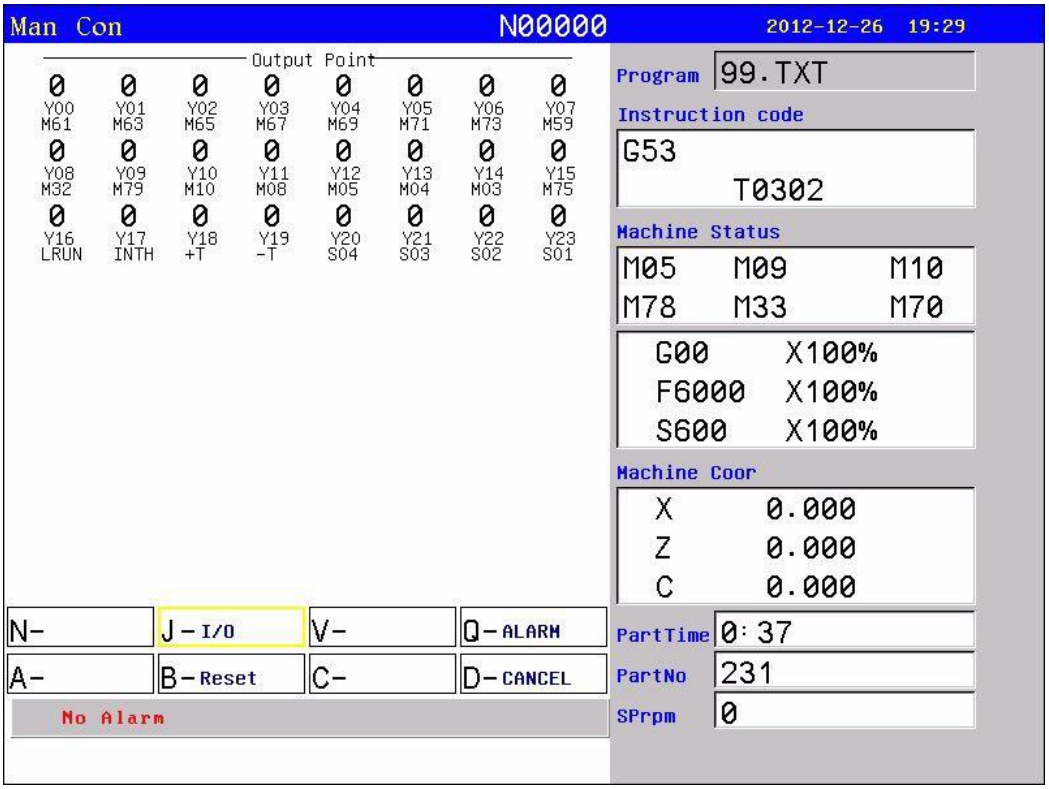
Compensation point and value contrast:



3.10 Diagnosis

Press “Diagnosis” to enter the Diagnosis Interface from the in Parameter Interface.
System Diagnosis Interface (Input signal)
Press “J” and “↑ ↓” to check the status of the Inputs and Outputs, or press “Q” to check alarm information.

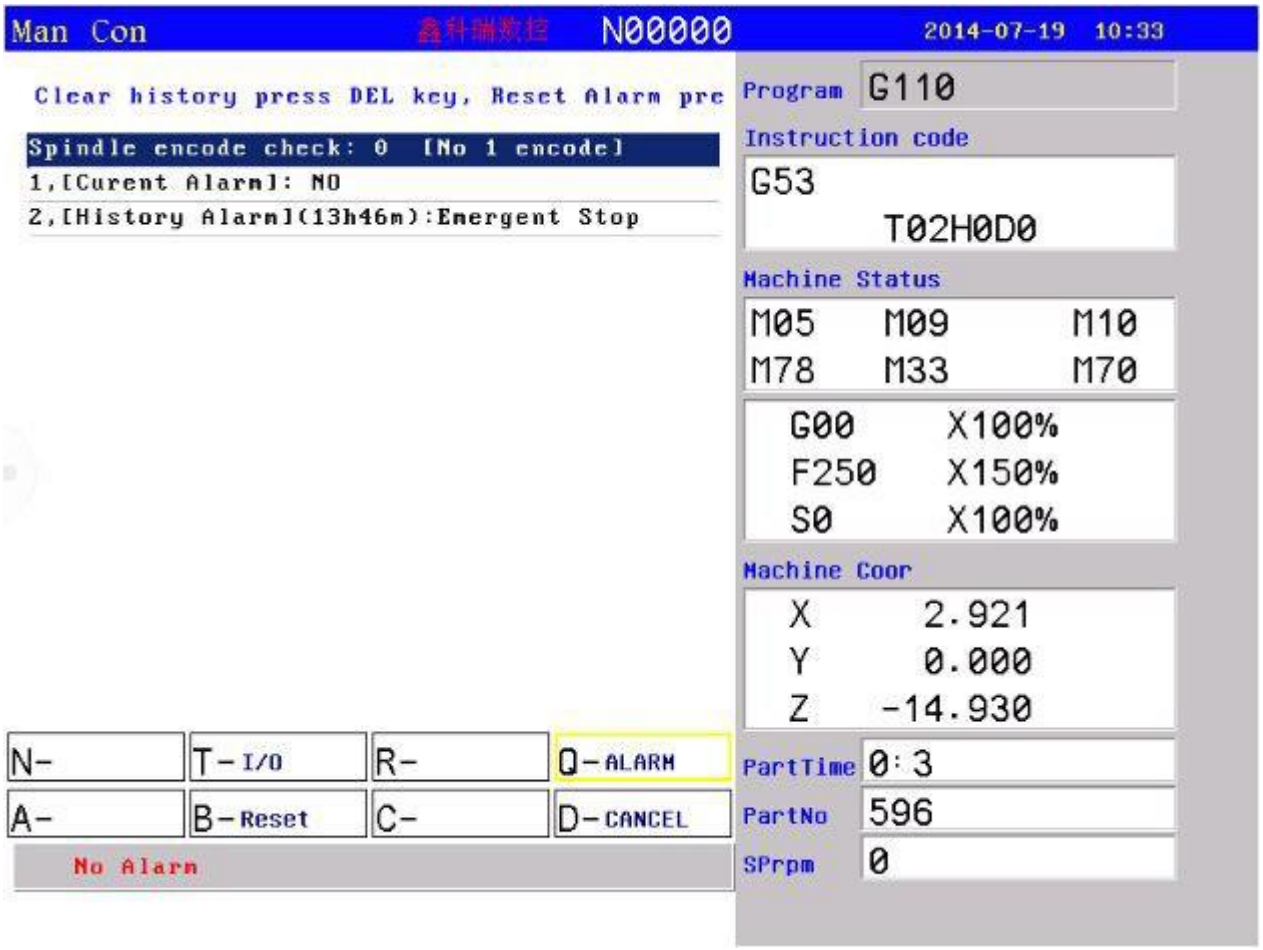




Checking interface of output signal

In the Input / Output Interface, 0 or 1 is the current status of the port.

1 means active, **0** means inactive.



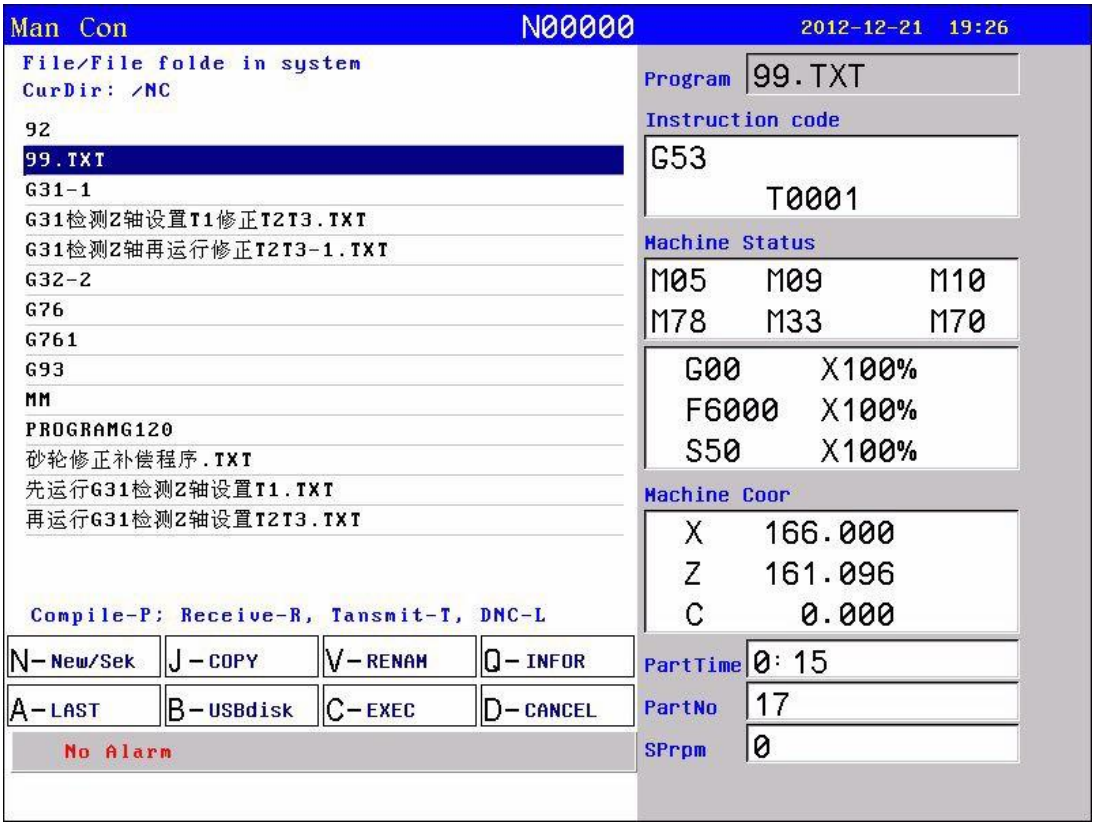
Alarm Information Interface:

The first line in this interface shows the number of spindle encoders, and any current or historical alarm information. Only the most recent ten (10) alarms are retained in memory, as any previous alarm messages are cleared automatically.

3.11 Program Operations:

Press “Program” in any menu to enter into the Programming status.

Program management is the same as file management. The program storage area is 32M bits, and there is no limit to the quantity of programs. Programming adopts full screen operation.



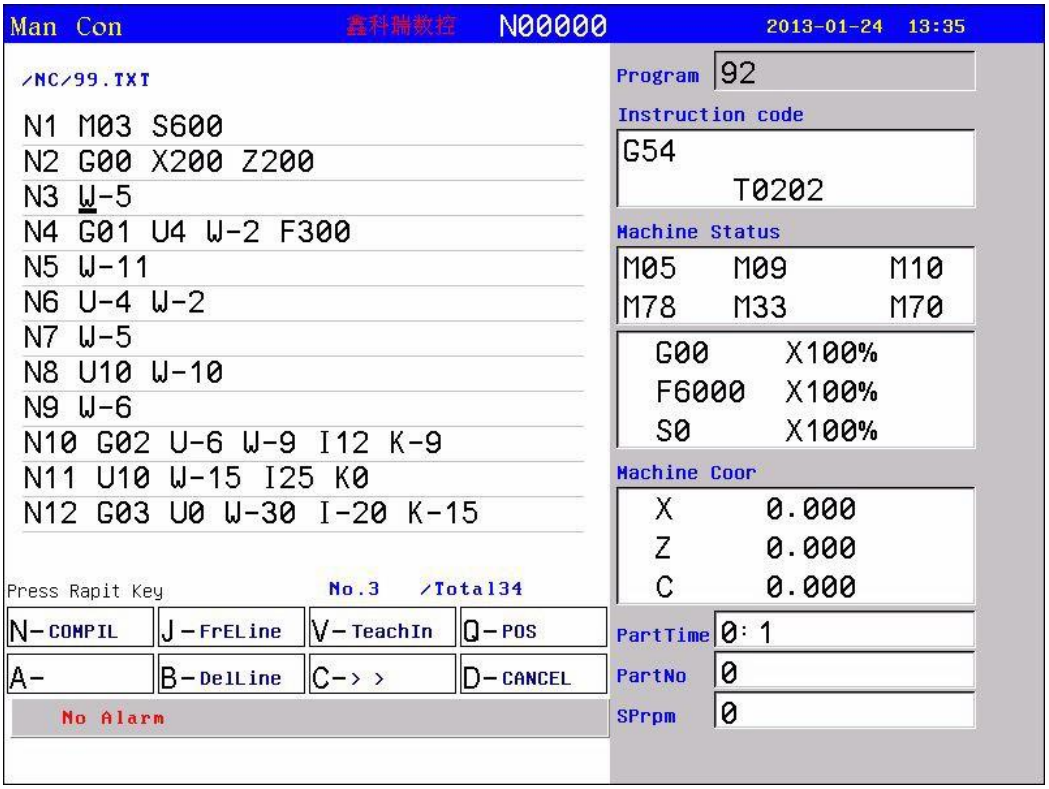
Center part of the screen is for program display with the current program displayed in reverse characters. Press “PgUp/PgDn” to choose a program, and press “Enter” to edit the current program. Function keys “N, J, V, Q, A, B, C, and D” correspond to “New file/search”, “Copy”, “Rename”, “Information”, Last grade”, USB disc”, “Execute program”, and “Cancel”.

3.11.1 Editing:

Select “New file/search” to popup a dialog box and input the name of program. If the program name exists, the program is called from memory. If the name does not exist, the system will build a new file.

The Program Name can be numbers, letters, or a combination up to 100 characters.

The system does not allow duplicate names. Build a new program, or select an existing program and press “Enter” to enter the editing interface.



The name of a program is displayed in the Prompt area in the top left corner of the screen. To the Left is the content of the program. Right is the status information for lathe. Operations in the Editing status are as follows:

(1) Locate the current cursor:

Press "↑ ↓ ← →" to move the cursor to any position in the program content.

Press "PgUp" to move the cursor to the previous page.

Press "PgDn" to move the cursor to next page.

(2) Modify character: Delete the character in the cursor position, and input any new characters directly.

(3) Insert character: Input new character directly at the cursor position. When a letter is input, a space is automatically produced in front. (When only a space is required, input a letter first, then delete the letter.)

(4) Delete character: Press "Del" in the cursor position.

(5) Inset a new line between existing lines: Press "Enter" to Insert a line ahead of the current line when the cursor is at the beginning of a line, otherwise, insert a line after the current line.

(6) "" Superimposing operation:

First page of function:

A. ""+N: Compile the current program.

B. ""+J: To the first line or last line of program.

C. “ ”+V: The function of determination, enter the status of handwheel; Press corresponding coordinate letters “X/Z/Y/C/A “ ” +Q (all coordinate axis)” to load coordinates for the current workpiece.

D. “ ” +Q: Locate to the appointed line.

E. “ ” +A: No operations.

F. “ ” +C: To select first page or second page of function.

G. “ ” +D: To switch between Chinese and English characters.

Second page of function:

A. “ ” +N: Delete the appointed part of program.

B. “ ” +J: Copy the appointed part of program.

C. “ ”+V: Sorting the program.

D. “ ” +Q: Search for the appointed character.

E. “ ” +A: Replace the appointed character.

F. “ ” +B: All the content replace the appointed character.

G. “ ” +C: To select first page or second page of function.

H. “ ” +D: To switch Chinese and character.

3.11.2 Copy:

Press “ ↑ ↓ ” in Program Main Interface to select a program which you need to copy. Press “J” to popup a dialog box to input a new name for the copied program. The new program will have the same content as the original, but under a different name so that you can modify, rename and back-up the copy.

3.11.3 Delete:

Press “ ↑ ↓ ” in Program Main Interface and select the program you wish to delete.

Press “Del” to delete the program.

(Caution: After deleting a program, it cannot be recovered.)

3.11.4 Rename:

Press " ↑ ↓ " in Program Main Interface to select the program you wish to rename.

Press "V" to popup a dialog box and input the new program name.

3.11.5 Information:

Press " ↑ ↓ " in Program Main Interface to select the program you wish to check. Press "Q" to popup a dialog box and check the size of the program and the remaining program space in the system.

3.11.6 Checking a program:

Press " ↑ ↓ " in Program Main Interface to select the program for checking. Press "P", and the system will check the form and grammar of the program, and display a prompt if a mistake is found.

3.11.7 Folder management:

You can build a file in this system, then press "N" in Program Main Interface to input a file name. Press "." to build a folder, and it will prompt you for a "Folder" name after the Program name.

Move the cursor to the file name and press "Enter" to open the file and build a new file, and if required, a new folder to place it in.

Press "A" go to the last folder.

To delete the Folder, move the cursor to the folder name and press "Del".

3.11.8 Select Automatic to run a program:

Press " ↑ ↓ " in Program Main Interface to select a program, and press "C" to select the program and switch to the last interface.

3.11.9 Program communications:

The system can use the RS232 serial port to deliver files.

Deliver (Transfers):

To transfer a program from this system to another system or save to a PC, press " ↑ ↓ " in Program Main Interface to select the program. Press "T" to deliver. Press "Reset" to interrupt the delivery process.

Receive:

To receive the selected program from another system or PC, it must be in text file form. Press "R", and input the name of the program to receive into the dialog box from the Program Main Interface. Press "Reset" to interrupt in the receive process.

Attention: 1. Use only the exclusive communication software to transfer a program to the User's PC.

2. The baud rates and comm settings of the system and the PC must be equal, or the transfer will fail.

3. The length of the RS232 cable must be no greater than 10 meters.

4. The serial port number must be the same as the system setting.

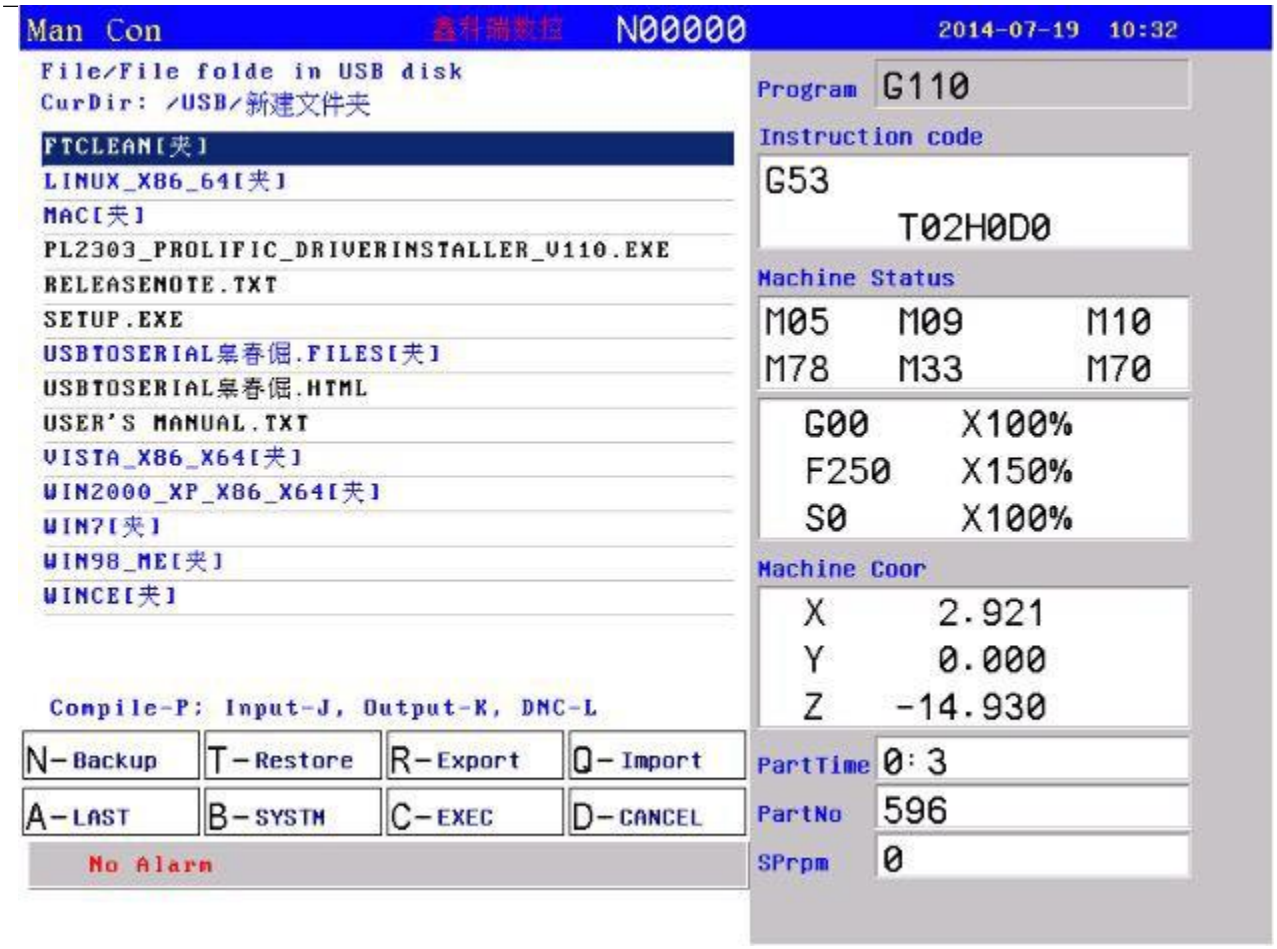
5. The PC program must be transferred in text file form.

3.11.10 U-disk management:

Parameter or System files can be transferred to another system or PC by U-disk. It may also be used to upgrade or back-up the software or system parameters.

Attention: The Folder Name must not include any "space" symbols.

Press "B" to enter the U-disk Management Interface from the Program Main Interface after inserting the U-disk into the USB port. Press "B" again to return to the System Interface.



A. The processing program management:

Copy the files or folder from U-disk into system:

After inserting the U-disk, press "B" to enter the U-disk directory from the Program Main Interface. Press " ↑ ↓ " to move the cursor and select a file or folder to copy.

Press "Q" to popup a dialog box and input a File Name. Press "Enter" to confirm. If there is a program of the same name in the system, a popup dialog box will ask if you wish to overwrite or recover the file or folder.

Copy the System files or folders to U-disk:

Press " ↑ ↓ " to move cursor and select a file or folder. Press "B", then press "V" to popup a dialog box and input the name in the U-disk Interface. Press "Enter" to confirm. If there is a program of the same name in the system, a popup dialog box will ask if you wish to overwrite or recover the file or folder.

Attention: Before unplugging the U-disk, return to the Program Files Directory Interface. Exit the U-disk Interface, or the data which was just copied will be lost.

The U-disk Name must not include any "space" symbols.

B. Using U-disk to manage parameters and system software:

The U-disk can be used to deliver files or system upgrade software, and to update back-up files and parameters. Method of operation is as follows:

Using the U-disk to copy parameters and software into the system. (Upgrades)

Insert the U-disk into the USB port and press “Program” to enter the Program Main Interface. Press “B” to display the files on the U-disk. Press “ ↑ ↓ ” to move the

cursor and select a folder to be copied into the system. Press “Enter” to open it, and press “J” to input code when the files appear. Press “Enter” to confirm, and wait for a few seconds for the files or parameters to be successfully copied. Press “B” to exit

U-disk after completing the operation, and restart the system. The system will reload with the new files and upgrade the parameter.

Attention: It is better to store the parameters in a separate folder on the U-disk to protect them from being accidentally erased or erroneously loaded.

To save or back-up parameter files using a U-disk:

Insert the U-disk into the USB port and press “Program” to enter the Program Main Interface. Press “B” to display the files on the U-disk. Press “N” to input the code, and press “Enter” to confirm. Wait for a few seconds for the process to complete. The system parameters are now on the U-disk. Press “B” to exit U-disk.

Attention: It is recommended that you extract the files onto an empty U-disk. (There are several dozen separate Parameter files). You should first create a new folder on your PC, then store these files there as a secure backup.

Chapter4 System installation and connection

4.1 System electrical specification

- 32bit high performance industrial grade ARM+DSP+FPGA
- 32M User's storage space
- 800x600 TFT LCD adaptive brightness, LED backlit LCD
- Touch type key board with excellent operational sensitivity.
- RS232 communication port
- USB port
- Photoelectric encoder
- 4 and 8 position electrical tool rest or tool post.
- High performance anti-interference switch-mode power supply.
- Variable frequency and speed governing for Two spindles
- Manual pulse generator
- Feed rate and spindle speed override controls

4.2 System technical index:

- Numerically Controllable X & Z axes:
- X & Z interpolation linked axes:
- Pulse equivalency: X axis with 0.001mm, Z axis with 0.001mm
- Maximum Axis Rapid Speeds: 30 Metres/min
- Maximum Feed Rates X & Z: 0.01-15,000mm/min
- Screw cutting: 0.1-1000mm in the metric system, 1-99 tooth/inch
- Minimum program input unit: 0.001mm
- Program coordinate range: ±99999.999mm
- Program coordinate system definition: IOS-841 international standard
- Program code: To IOS-840 international standard
- Mean Time Between Failure (MTBF): Greater than 6,000 hours

4.3 Operating Environment:

- Power supply: AC 220V (+10%,-15%), frequency 50Hz ±1%

- Power source≤100W

- Operating temperature: 5~45℃, relative humidity: 40~80%
- Storage and transportation temperature: 0~55℃, relative humidity less than 90%(40℃)

4.4 System installation and connection

At first, users should check whether the hardware is complete, undamaged and compatible, such as: CNC system, driving power, servo motors, photoelectric encoders, and electric tool carrier. (Tool Turret)

The installation of CNC system must be securely installed, with some space between components to ensure good air ventilation. Control Panel should be installed in a place where it is convenient to operate and not easily damaged by foreign objects. The mains power supply must be stable and within manufacturers specifications.

CNC system and drivers must be kept well away from machines which operate at high currents. In order to reduce interference, all signal cables should be kept away from AC contactors. Photoelectric encoders, limit switches, and basic point signals must be kept separate from high current cabling, like Servo Motor power cables and solenoids. All power cords must be suitably grounded.

Secure all plugs with screws. Never insert or extract cables when power is on.

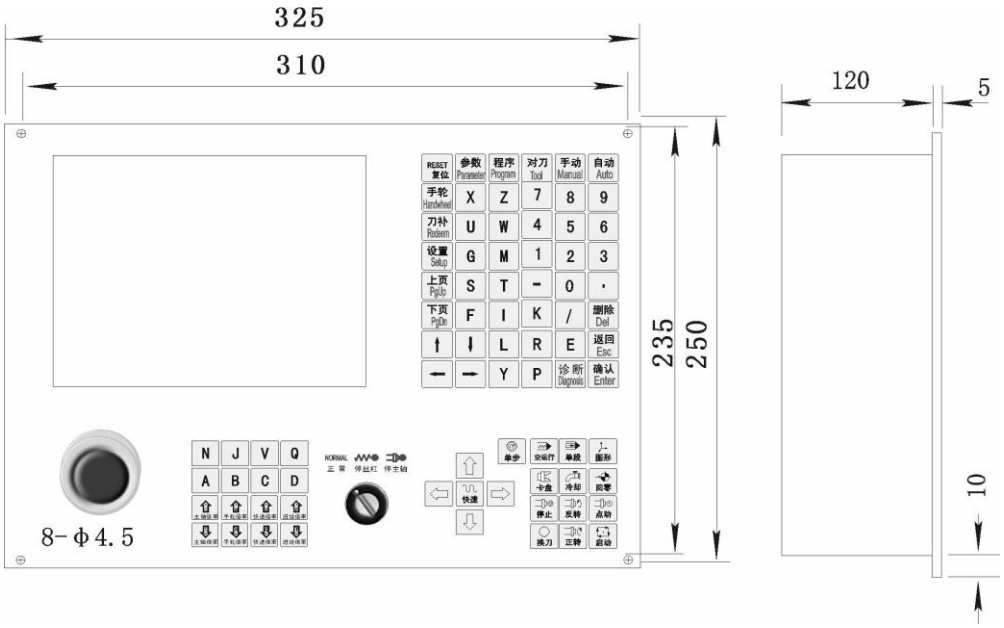
When installing the CNC system, ensure that the panel is not in danger of being damaged by hot, hard, or sharp items. If any painting of other parts of the machine is planned, please remove the CNC control panel to keep it clean.

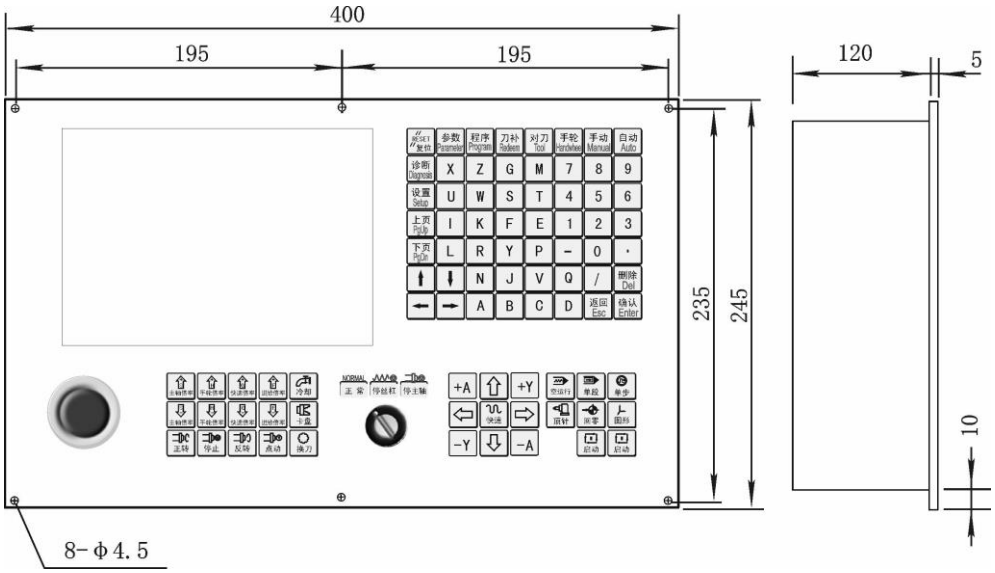
Ensure that strong magnetic fields and high current equipment will not cause interference problems with the system. Keep well away from flammables, explosives, and other dangerous materials.

Attention:

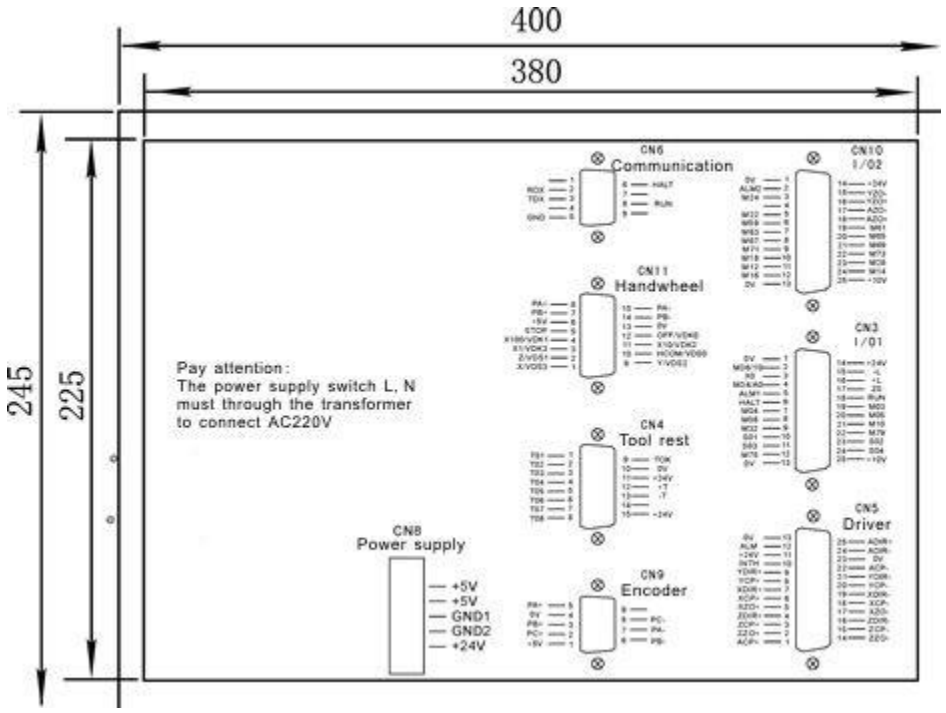
1. **All of the equipment must be housed in an electrical cabinet which has good lightning protection.**
2. **The installation must be secure to avoid damage from vibrations.**
3. **Do not install near flammable goods or any high heat sources.**

4.5 System installation dimension (Type-1: 325x250x120; Type-2: 400x245x120)





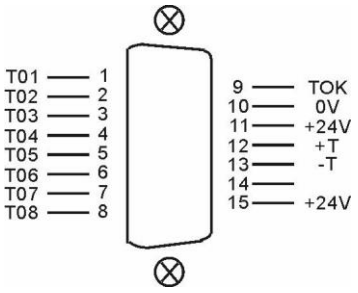
4.6 System rear view



Attention: The Switching power supply Live & Neutral must be fed through an isolation transformer capable of AC 220V, @ current 0.5A.

4.7 Interface connection graph

4.7.1 CN4 and electric tool rest connection



CN4 DB15F (Socket) Electric Tool Carrier (Turret)				
Signal	Pin	I/O	Function	Availability
0V	10	OUT	Grounding	0V
+24V	11、15	OUT	+24V Power supply	+24V
+T	12	OUT	Forward rotate signal	0V
-T	13	OUT	Reverse rotate signal	0V
T1	1	IN	T1 signal	0V
T2	2	IN	T2 signal	0V
T3	3	IN	T3 signal	0V

T4	4	IN	T4 signal	0V
T5	5	IN	T5 signal	0V
T6	6	IN	T6 signal	0V
T7	7	IN	T7 signal	0V
T8	8	IN	T8 signal	0V
TOK	9	IN	Lock signal	0V

This system can store data for 1-99 tools. Control Signals are available for a maximum of eight (8) tools in an electric turret. (Default is 4 tools). Press "C" to set the total tools in "Redeem". The starting tool is set in the Tool parameters.

Parameter No.1: Activate the function of electrical tool to select electrical tool or tool post. [**1** means Yes, **0** means No]

Parameter No.2: Activate tool number when using electrical change into tool post, if the electrical tool is 4, set the number is 5 to switch to tool post.

Parameter No.4: The maximum time for positive tool rotation. If the required tool is not found within this time period, the rotation will stop, and alarm. (Unit: second)

Parameter No.5: The Delay time after a positive tool rotation to verify the tool position signal. (ms)

Parameter No.6: Delay time after the tool change is complete. (ms) Parameter No.7: Clamping time after tool reverse rotation (ms).

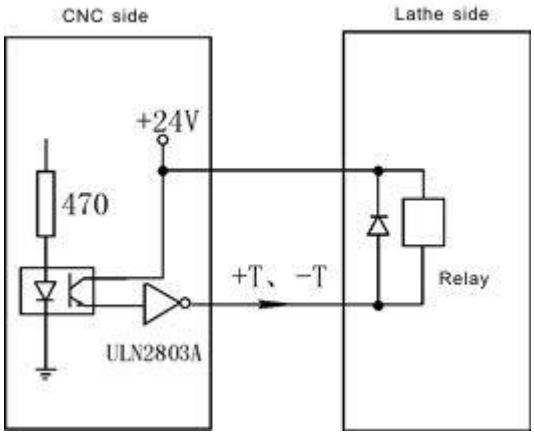
Parameter No.9: Tool clamped signal. Checks the tool turret locked switch status after a tool reverse rotation. (Signal TOK) Default is "Not checked".

Attention:

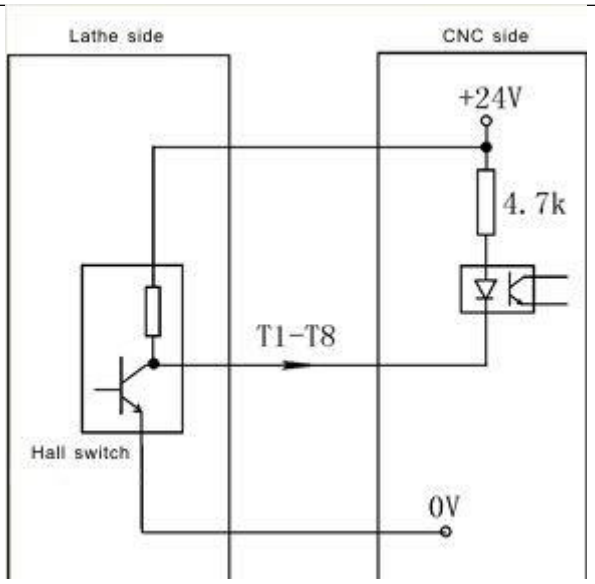
1. All of the inputs and outputs are for the system. Inputs are from external sources to the system. Outputs are from system signals to external devices.

2. When choosing electrical interface modules, +T and –T control a single contact centre-off relay. The user should install two AC contactors for +T and –T.

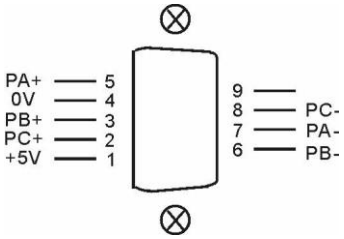
System output signal +T –T:



Tool inputs T1~T8 & TOK:



4.7.2 CN9 Spindle Encoder connections:



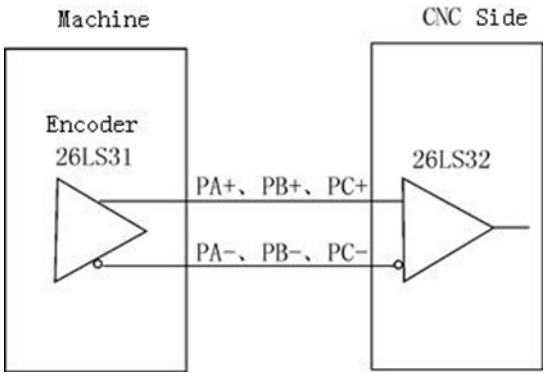
CN9 DB9(pin) spindle encoder				
signal	pin	I/O	function	availability
0V	4	OUT	0V	0V
+5V	1	OUT	+5V	+5V
PA+	5	IN	+A signal	5V
PA-	7	IN	-A signal	
PB+	3	IN	+B signal	5V
PB-	6	IN	-B signal	
PC+	2	IN	+Z signal	5V
PC-	8	IN	-Z signal	

Attention:

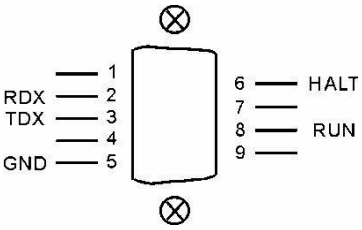
1. Output signals from the encoder are TTL compatible line outputs, supplied from +5V.

2. The signal line must be a shielded twisted pair cable, of 20M max length.

Input signals PA PB & PC from the encoders:



4.7.3 CN6 Computer System connections:



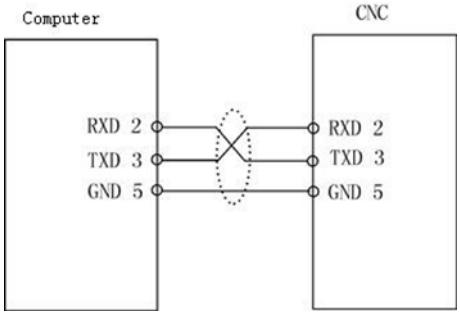
CN6 DB9F (Socket) RS232 communication				
Signal	**Pin**	**I/O**	**Function**	**Availability**
0V	5	OUT	0V	0V
RXD	2	IN	RXD	
TXD	3	OUT	TXD	
RUN	8	IN	run	0V
HALT	6	IN	pause	0V

Attention:

1. The exclusive system communication software must be used to connect an exterior PC for data communication.

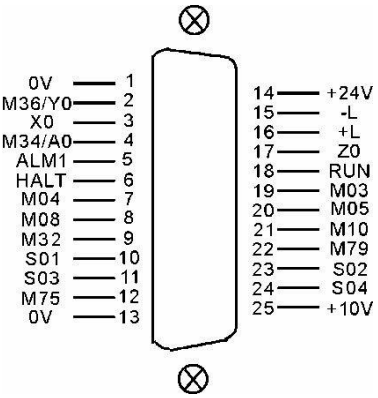
2. The signal line must be a shielded twisted pair cable, length of 10m max.

Connecting CN6 signals to a PC:



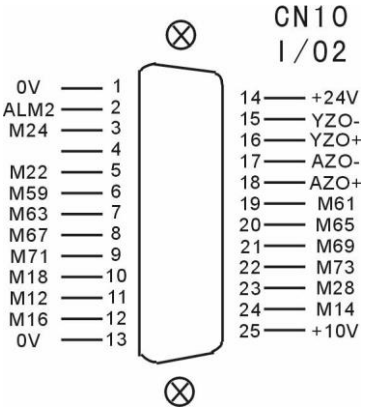
Attention: When programming on a PC, all files must be saved as text files.

4.7.4 CN3 Machine Electrical device I/O-1 connections:



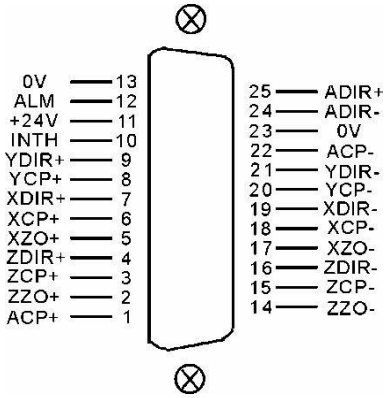
CN3 DB25F (Socket) I/O-1 Machine Signals				
Signal	**Pin**	**I/O**	**Function**	**Availability**
0V	1	OUT	0V	0V
+24V	14	OUT	+24V	+24V
M36/Y0	2	IN	M36/Y0	0V
X0	3	IN	X axis Zero	0V
Z0	17	IN	Z axis Zero	0V
-L	15	IN	Positive limit	0V
+L	16	IN	Negative limit	0V
M34/A0	4	IN	M34/A0	0V
ALM1	5	IN	Transducer alarm-1	0V
HALT	6	IN	Pause	0V
RUN	18	IN	Run	0V
M03	19	OUT	Spindle clockwise	0V
M04	7	OUT	Spindle counter-clockwise	0V
M05	20	OUT	Spindle stop	0V
M08	8	OUT	Coolant On	0V
M10	21	OUT	Spindle chuck Clamp	0V
M32	9	OUT	Lubrication On	0V
M79	22	OUT	Tailstock Extend	0V
S01	10	OUT	Spindle first gear	0V
S02	23	OUT	Spindle second gear	0V
S03	11	OUT	Spindle third gear	0V
S04	24	OUT	Spindle fourth gear	0V
M75	12	OUT	C axis mode	0V
+10V	25	OUT	Primary Spindle Analog	0～10V
0V	13	OUT	0V	0V

4.7.5 CN10 Machine Electrical device I/O-2 connections:



CN10 DB25F (Socket) I/O-2 Machine Signals				
Signal	**Pin**	**I/O**	**Function**	**Availability**
0V	1	OUT	0V	0V
+24V	14	OUT	+24V	+24V
ALM2	2	IN	Machine alarm-2	0V
M24	3	IN	M24	0V
M22	5	IN	M01 input	0V
M59	6	OUT	Huff	0V
M61	19	OUT	M61	0V
M63	7	OUT	M63	0V
M65	20	OUT	M65	0V
M67	8	OUT	M67	0V
M69	21	OUT	M69	0V
M71	9	OUT	M71	0V
M73	22	OUT	M73	0V
M18	10	IN	M18	0V
M28	23	IN	M28	0V
M12	11	IN	M12	0V
M14	24	IN	M14	0V
M16	12	IN	M16	0V
YZO+	16	IN	+Y Servo Zero signal	5V
YZO-	15	IN	-Y motor Zero signal	
AZO+	18	IN	+A motor Zero signal	5V
AZO-	17	IN	-A motor Zero signal	
+10V	25	OUT	the second spindle converting	0～10V
0V	13	OUT	0V	0V

4.7.6 CN5 Servo Drive and Motor connections:



CN5 DB25(pin) servo drive signal				
signal	pin	I/O	Function	Availability
XCP+	6	OUT	X Pulse signal +	5V
XCP-	18	OUT	X Pulse signal -	
XDIR+	7	OUT	X Direction signal +	5V
XDIR-	19	OUT	X Direction signal -	
YCP+	8	OUT	Y Pulse signal +	5V
YCP-	20	OUT	Y Pulse signal -	
YDIR+	9	OUT	Y Direction signal +	5V
YDIR-	21	OUT	Y Direction signal -	
XZO+	5	IN	X motor Zero +	5V
XZO-	17	IN	X motor Zero -	
ZCP+	3	OUT	Z Pulse signal +	5V
ZCP-	15	OUT	Z Pulse signal -	
ZDIR+	4	OUT	Z Direction signal +	5V
ZDIR-	16	OUT	Z Direction signal -	
ZZO+	2	IN	Z motor Zero +	5V
ZZO-	14	IN	Z motor Zero -	
ACP+	1	OUT	A Pulse signal +	5V
ACP-	22	OUT	A Pulse signal -	
ADIR+	25	OUT	A Direction signal +	5V
ADIR-	24	OUT	A Direction signal -	
0V	13、23	OUT	0V	0V
ALM	12	IN	Servo alarm	0V
+24V	11	OUT	+24V	24V
INTH	10	OUT	Clear alarm	0V

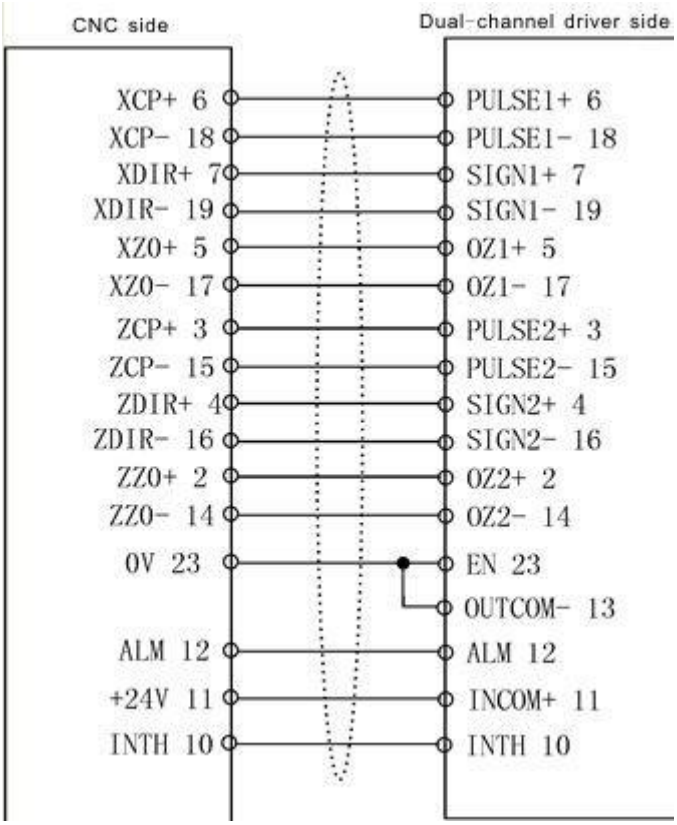
Attention:

1. The drive signal line must be shielded twisted pair cable, of 20m max.

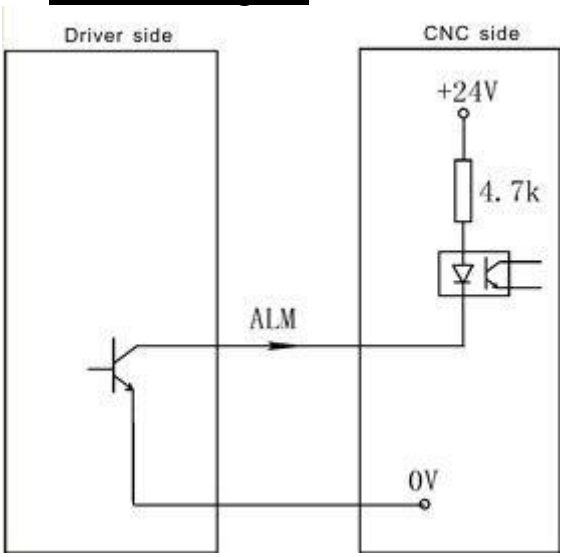
2. The alarm signal ALM is set to be either Normally Open or Normally Closed by Other Parameters No.17.

3. When the C axis is defined as a Rotating Axis, an M800 instruction "Homes" the axis to the datum (Zero) point of the encoder. An M75 input signal selects the Position Control mode for the Spindle Servo. M03/M04 signals cancel the M75 signal, returning the spindle servo to the Speed Control mode.

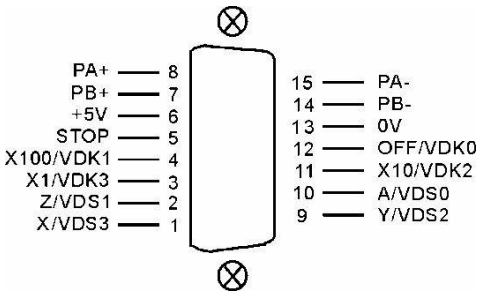
CN5 X Z axis connects to dual-channel servo driver CN3:



Servo alarm signal:



4.7.7 CN11 Handwheel, Step Switch and External Control Switch connections:



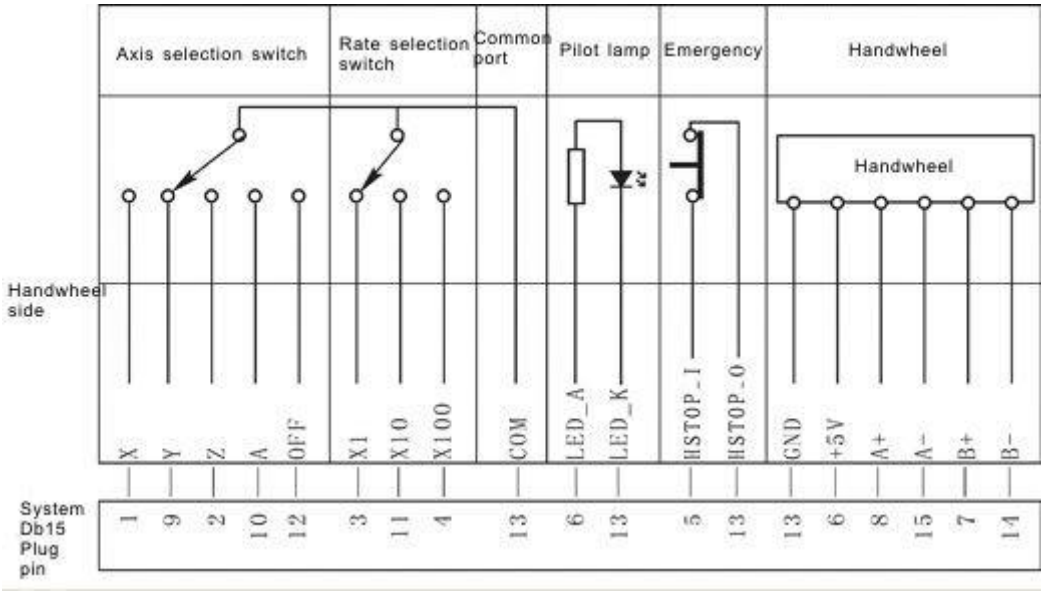
CN11 DB15M (Plug) Hand wheel, Step switch connections				
Signal	**Pin**	**I/O**	Function	Availability
0V	13	OUT	0V	0V
+5V	6	OUT	+5V	+5V
PA+	8	IN	A signal +	5V
PA-	15	IN	A signal -	
PB+	7	IN	B signal +	5V
PB-	14	IN	B signal -	
STOP	5	IN	EMERGENCY STOP	0V
OFF/VDK0	12	IN	Off / Feed Hold	0V
X100/VDK1	4	IN	*100/ Feed Step 1	0V
X10/VDK2	11	IN	*10/ Feed Step 2	0V
X1/VDK3	3	IN	*1/ Feed Step 3	0V
A/VDS0/HALT	10	IN	A / SP amending 0 / SP Stop	0V
Z/VDS1	2	IN	Z / SP amending 1	0V
Y/VDS2/RUN	9	IN	Y / SP amending 2 / SP RUN	0V
X/VDS3	1	IN	X / SP amending 3	0V

4.7.7.1 Electrical handwheel (Manual pulse generator):

When Other Parameters No.1 is set to "**1**", a standard external handwheel may be connected, but external step switches cannot be used to adjust spindle speeds, feed rates, and external Run / Stop buttons. Axis Parameters No.1 & No.2 must be set to "**0**" for this to be enabled. To use A, X, Y, Z, X1, X10, X100 controls, Other Parameters No.33 No.34 must be set to "**0**".

PA+ PB- PA+ PA- Input signal from Handwheel Pulses A & B.

Handwheel connection diagram:



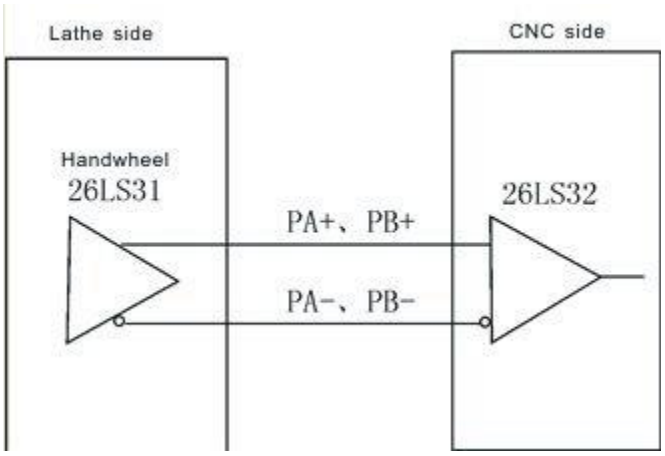
Attention:

1. The output signals from the handwheel are standard TTL line outputs. The Supply Line voltage is +5V.

2. Just connect PA+ PB+ if only a voltage output is required.

3. If the Manual Pulse Generator does not require an "Enter" button. If an "Enter" button is fitted, use the line to short the ends of switch.

Input signals from Handwheel:



4.7.7.2 Using the Band (Step) Switch:

When Axis Parameters No.1 & No.2 are set to "**1**", the Band (Step) switch is active, and the handwheel cannot be used as external switches or an external Run button. If Other Parameters No.1 is set to "**0**", and Other Parameters No.33 & No.34 are also set to "**0**", A, Z, Y, X, OFF, X100, X10, and X1 are then used to select the Feeding axis and Step Increments.

VDS0(A), VDS1(Z), VDS2(Y), VDS3(X), are the Input Signals used to adjust the spindle speed, in a total of 16 steps. VDK0(OFF), VDK1(X100), VDK2(X10), and VDK3(X1) are Input Signal for adjusting Feed Rates, in a total of 16 steps.

4.7.7.3 External Run / Stop button:

When Other Parameters No.33 is set to "**1**", Pin #9 of CN11 can be connected to an external Run button, and used to Start an automatic program. If Other Parameter No.34 is set to "**1**", Pin #10 of CN11 can be connected to an external Stop button to Stop the program.

4.7.7.4 External EMERGENCY STOP:

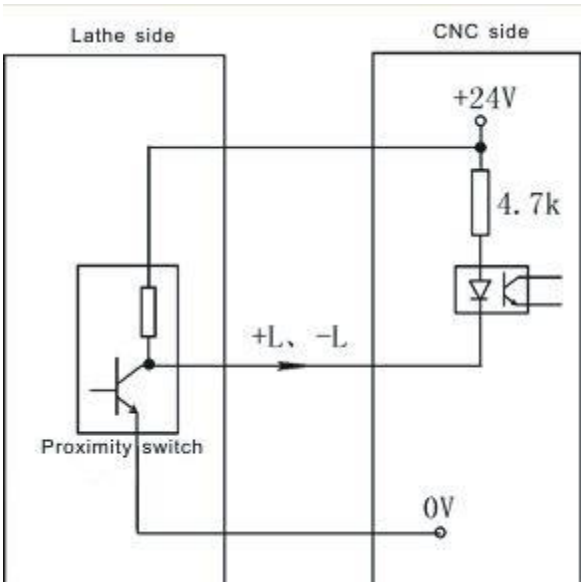
The STOP signal is the input used for an external EMERGENCY STOP button. Other Parameters No.27 controls the Emergency Stop signal as either Normally Open, or Normally Closed. It is recommended for safety reasons to always set this as Normally Closed.

4.8 Installation on a Lathe.

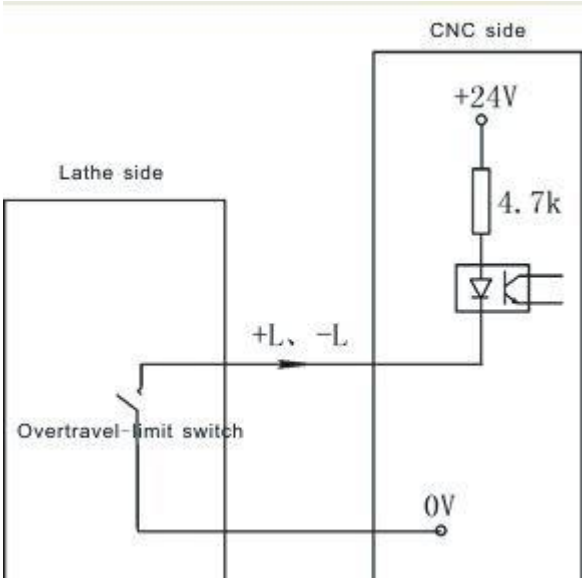
4.8.1 Limit Positions: Set X & Z axes Positive Limits:

Example:

Model 1: NPN approach switch



Mode 2: general switch



Axis Limit Switch Parameters:

No.21, X& Z Positive Limits, [“**0”** normally open, “**1”** = normally closed] No.22, X & Z Negative Limits, [“**0”** normally **o**pen, “**1”** = normally closed]

Attention:

1. X & Z Axis Limit Switches share common Normally Open or Normally Closed input signals. Positive limit and negative limit signals are indicated by +L and –L signs.

2. Please consider using our Electrical Interface Modules.

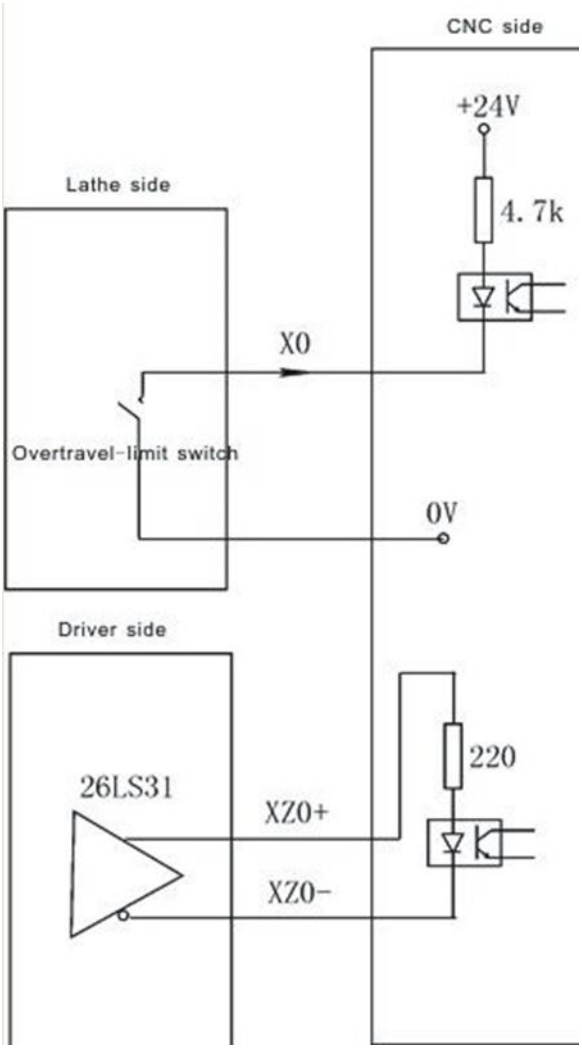
3. The system can define X0 Z0 as the limit input signal for the X & Z axes. By using the X0 signal as the limit signal and the Home datum point of the X axis, the same switch can be used to control both. The same can be applied to the Z0 limit signal and Z axis Home datum point.

To use the same switch to control both, set Parameters No.21 & No.22 as follows:

No.21, X axis limit position ["**0**" = normally open, "**1**" = normally closed]
No.22, Z axis limit position ["**0**" = normally open, "**1**" = normally closed]
(Functions must be copied with our exclusive PLC software.)

4.8.2 Lathe's datum point (reference point or 0 point):

Using the X axis as an example. (The same applies for the Z axis)



Returning to the datum point:

If the Floating Datum Point function setting is not used or invalid, returning the machine axes to the datum point (Homing) is required, to check the Home switch signal and motor Z pulse signal. No.23 parameter in axis parameter is set to be "00000000".

Axis Parameter No.26 is used to set the Return to Datum Point functions:

There are four different methods for returning to or setting the datum point when the system is powered on:

When Axis Parameter No.26 is set to "**1**": Open Mode: The system does not prompt, and no limit switches or extra activities are required.

When set to "**0**": Prompting Mode: A popup dialog box will prompt the user to Home the axes to the datum points. No limits are set.

When set to "**8**": Forced Mode: A dialog box will popup, forcing the user to Home the axes, before further processing or running automatically. The system will display "**Feeding axis doesn't back to the datum point**" and will not process any programs.

When set to "**9**": Super Forcing Mode: A popup dialog box will display "**Feeding axis doesn't back to the datum point**". All axis feeding and other processes are inhibited until the machine has been Homed.

Axis Parameter No.27 sets the checking signal mode for Homing to the datum point:

When Axis Parameter No.27 is set to "**0**": After contacting the datum point switch, the servo will slowly reverse off of the switch, and stop when it receives the "0" pulse signal from the motor encoder.

When set to "**1**": After contacting the datum point switch, the servo will slowly reverse off of the switch, and stop immediately when off the switch.

When set to "**2"**: After contacting the datum point switch, the servo will continue forward motion until it moves off the switch, and stop when it receives the "0" pulse signal from the motor encoder.

When set to any other value: After contacting the datum point switch, the servo will slowly continue forward motion until it moves off the switch, and stop immediately when off the switch.

The Datum Point Homing mode should be set to best suit the conditions and equipment. Under most common situations, we suggest that modes 0 or 2 be used, because if the "0" pulse signal of the motor encoder is not referenced, the accuracy of the Home Datum Points cannot be guaranteed.

Axis Parameter No.28 sets the direction and sequence for returning to the datum point:

Each axis is set separately. Positional Parameter D2 controls the Home return direction of the X axis, and D4 controls the return direction of the Z axis. "**1**" means move in the negative direction, "**0"** means move in the positive direction. D8 controls the sequence of X and Z. "**1"** means move Z first, "**0"** means move X first.

Axis Parameter No.29 sets the type of datum point switch:

Each axis is set separately. Positional Parameter D0 controls the X axis, D2 controls the Z axis. ["1" = normally closed, "0" = normally open]

Axis Parameters No.30 & No.31 are set to check the processing length of the axis motor Zero pulse after returning to the datum point:

This is used to set the distance required to check for the motor encoder zero pulse signal after the axis moves back off of the axis datum switches X(No.30), and Z(No.31) Unit: 0.1mm.

Attention: The parameter value must less than the distance the motor turns one revolution, or incorrect datum point data could be set.

Speed Parameters No.31 No.33 set the speed of approach towards the datum point switch when Homing the axis. (Zero point):

The maximum axis return speed to the datum point switch, when X(No.31) & Z(No.33) are set for returning to a datum point. [Range 20-500 mm/min.]

Attention: This parameter value influences the accuracy of the Home datum point. A smaller value results in a slower the return speed, but greater accuracy. Once this value has been set, do not change it, or the reference point may also be affected.

Axis Parameters No.32 & No.33 set the offset distance for finishing back to the datum point:

These parameters reset the machine coordinates to the offset distance from the reference point to the Zero Datum Point, after each axis is returned to the Home position and the zero pulse signal of the servo motor is received.

[Numeric range: -99999 to +99999, in 0.01mm Units.]

This parameter value is directly related to the distance between the Home datum point, the Zero datum point of the lathe, and the lathe coordinates.

Attention: After returning an axis to the datum point, the offset speed is determined by G00.

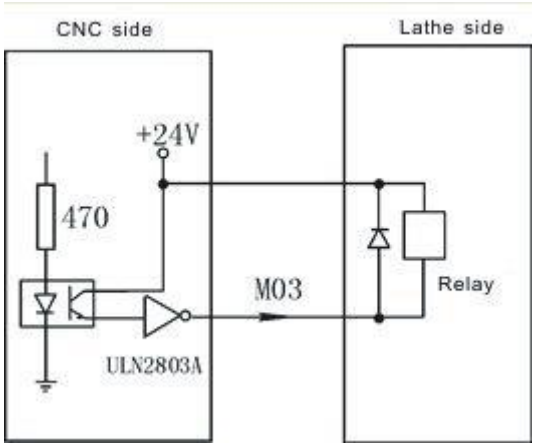
1. NPN devices can also be used as speed reducing switches.

2. The axis speed reduction distance must also be considered when installing the speed reducing / Home switch.

4.8.3 Control Signal switches:

Using M03 as an example: (It is the same for M04 M05 M08 M10 M32 M79 M75 M59 M61 M63 M65 M67 M69 M71 M73 S01—S04)

M03 signal control:



As in the above diagram, a return circuit is formed with a 24V source when the system M03 output is activated. An intermediate relay is used as a normally open contact to form a circuit with the Spindle Rotation AC contactor.

All of the signals are active low level (0V) outputs.

Attention:

When these outputs are connected to relays and inductive loads, diodes must be added to prevent system damage from reverse currents. If electromagnetic contactors are used, fit resistive and capacitive spark absorbing devices.

4. Output ICs (ULN2803A) and corresponding output ports:

a). U28 controls M59, M61, M63, M65, M67, M69, M71, & M73.

b). U29 controls M03, M04, M05, M08, M10, M79, M32, & M75.

c). U30 controls +T, –T, S01, S02, S03, S04, LRUN, & INTH.

5. When user-defined signal M71/M70, and M73/M72 are used for input signals from spindle chuck and tailstock, they cannot be set as "user-defined".

Other Parameters No.20 & No.21.

6. When user-defined signal M65, M67, & M69 are used for "Emergency Stop" input signals, they cannot be set as "user-defined".

7. Set Other Parameters No.28 & No.29.

4.8.3.1 System Primary Spindle Control (M03/M04/M05)

This System has outputs for two spindle control signals. (First spindle, second spindle), as follows:

Axis parameters:

No.7: Set the spindle braking time, also the Holding Time output. The less time, the faster the braking.

No.8: Set the braking signal as a long signal "**1**" or as a short signal "**0**".

No.9: To check the spindle position from the Spindle encoder signal feedback. ["**1**" =check, "**0**" =do not check.

No.10: Set the spindle encoder feedback pulses per rotation. (Encoder Lines x 4)

No.50: Rotate the spindle when changing gears. ["1" =Yes, "0" = No]

No.51: Spindle motor speed for changing gears. (1/100rpm)

No.52: Spindle direction when changing gears ("0" =positive, "1" =negative)

No.53: The stopping time when spindle shifting (10ms)

No.54: Rotating time at low speed when spindle shifting (10ms)

No.55: Stopping delay time of spindle (10ms)

Speed parameters:

No.8: Spindle speed in Manual mode. Unit: rpm.

No.36: Maximum Spindle speed at 10V Analog Output voltage.

No.37: Highest speed of the Primary Spindle in low gear, or the highest speed of the Secondary Spindle, at 10V Analog Output voltage. Unit: rpm.

??? (Second Gear)???

No.38: The highest speed of the Primary Spindle in Third gear, at 10V Analog Output voltage. Unit: rpm

No.39: The highest speed of the Primary Spindle in Fourth gear, at 10V Analog Output voltage. Unit: rpm

No.40 parameter: The highest speed of the Secondary spindle, at 10V Analog Output voltage. Unit: rpm

Other parameters:

No.13: Set to interlock the spindle and chuck: "**0**" means they are separate. "1"

means the spindle can only start turning when the chuck is clamped. The tailstock cannot be operated while the spindle is rotating.

This parameter setting is directly related to the configuration of the lathe and to the user's service conditions, but a setting of "**1**" is suggested for safety reasons.

4.8.3.2 System Lubrication Control (M32/M33):

Other Parameters No.4 enables the automatic lubrication function. Parameter No.6 sets the period between lubrication events (Unit: S), and Parameter No.5 sets the lubrication activity time (Unit: S).

4.8.3.3 Chuck and Tailstock control (M10/M11 M79/M78):

Chuck and tailstock control parameters for this system are as follows:

Other parameters:

No.2: Sets the controls for the type of chuck, and if the clamping is for internal or external workpieces. (Inner: Chuck clamps to center when M10; Outer: Chuck clamps outward when M10). ["**1**" means outer, "**0**" means inner.]

No.15: Check if the chuck is clamped when under automatic control. "**1**" means check M12 when M10 is active, and check M14 when M11 is active; "**0**" means do not check.

No.16: Check if the tailstock is extended or retracted when under automatic control. "**1**" means check M18 when M79 is active, and check M28 when M78 is active; "**0**" means do not check.

No.20: Sets the Chuck Control signal to toggle between On and Off. (Only One signal is required for a one-way valve or two signals for a two-way valve). This parameter depends upon the equipment on the lathe.

M10 is a single output signal to control the chuck clamping. When this parameter is set to "**0**", and M10 is active, the chuck will clamp. When M10 is inactive, the chuck will unclamp.

M10 and M71 clamp and unclamp the chuck when set to "**1**". The system will clamp the chuck when relay M10 is active, and M71 is inactive. The chuck will unclamp when M10 is inactive and M71 is active. Program M10 for Chuck Clamp, and M11 (Output M71) for unclamp.

No.21: Controls the tailstock from only one signal (one-way valve), or two signals (two-way valve). This parameter depends upon the equipment on the lathe.

[If set to "**0**", the output will toggle. M79 Controls the tailstock activity. When M79 is active the tailstock will extend. When inactive, the tailstock will retract.]

[If set to "**1**", an M79 command extends the tailstock, and M78 retracts it. When M79 is inactive, and M78 is active, Output M73 will retract the tailstock.

No.22: If set to "**0**", an M16 command causes the chuck clamping signal to toggle between On and Off. If set to "**1**", the chuck clamping is controlled by a push button or a foot switch

No.23: If set to "**0**", an M14 command causes the tailstock extend signal to toggle

between On and Off. If set to "**1**", the tailstock is controlled by a push button or a foot switch

No.24: Sets the retention time for an M10, M71 chuck clamp signal. Unit: Secs.

No.25: Sets the retention time for an M79, M73 tailstock extend signal. Unit: Secs.

Attention: M12, M14, M16, M18, and M28 are multiple functions signals. Select only one function to use for each.

4.8.4 System alarm signals:

(ALM, ALM1, ALM2, Door alarm M12, and EMERGENCY STOP)

Other parameters:

No.7: Checks the Protective Door switch. Set to "**0**" if no door switch is required. Set to "**1**" to check if the door is closed before automatic operation. For safety reasons, setting to "**1**" is highly recommended.

No.8: The type of door switch is "**1**" =Normally Open, or "**0**" =Normally Closed.

No.17: Sets the servo alarm signal (Pin 12 of CN5 [ALM]) as "**0**" =Normally Open,

or "**1**" =Normally closed.

No.18: Sets the spindle alarm signal (Pin 5 of CN3 [ALM1]) as "**0**" =Normally Open, or "**1**" =Normally Closed.

No.19: Sets the main alarm signal of the lathe (Pin 2 of CN19 [ALM2]) as "**0**" =Normally Open, or "**1**" =Normally Closed.

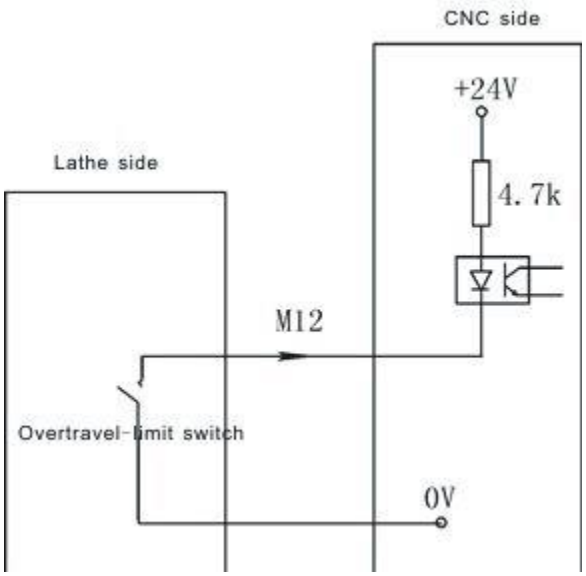
No.27: Sets the EMERGENCY STOP signal of CN11 as "**0**" =Normally Open, or "**1**" =Normally Closed. For safe operation, Normally closed is highly recommended.

EMERGENCY STOP: Press the "Emergency" Stop button if an accident occurs. The lathe will stop all activities, and the system screen will display the "Emergency" message. Only release the EMERGENCY STOP button when the problem is identified and made safe. The M67 Output M67 signal is activated (Alarm Output) when Other Parameter No.29 parameter is effective. (This output signal can be used to protect the lathe by cutting off the power supply).

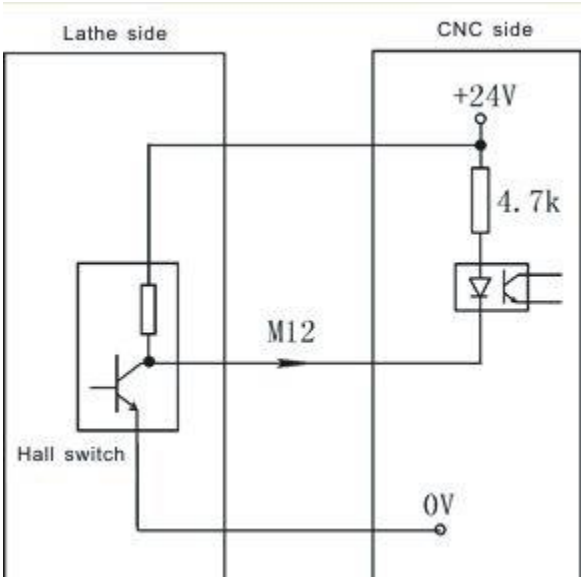
4.8.5 User-defined output signal M12:

(M14 M16 M18 M28 M22 M24, external "Run", external suspend "HALT", and external "STOP" are the same.)

M12 Output wired into over-travel limit switch wiring:



M12 switch can also operate as a NPN checking witch:



Attention:

1. M12, M14, M16, M18, and M28 are all multiple functions signals. It is only possible to use one for each function at any one time.

2. All of these signals are "Active Low" (0V).

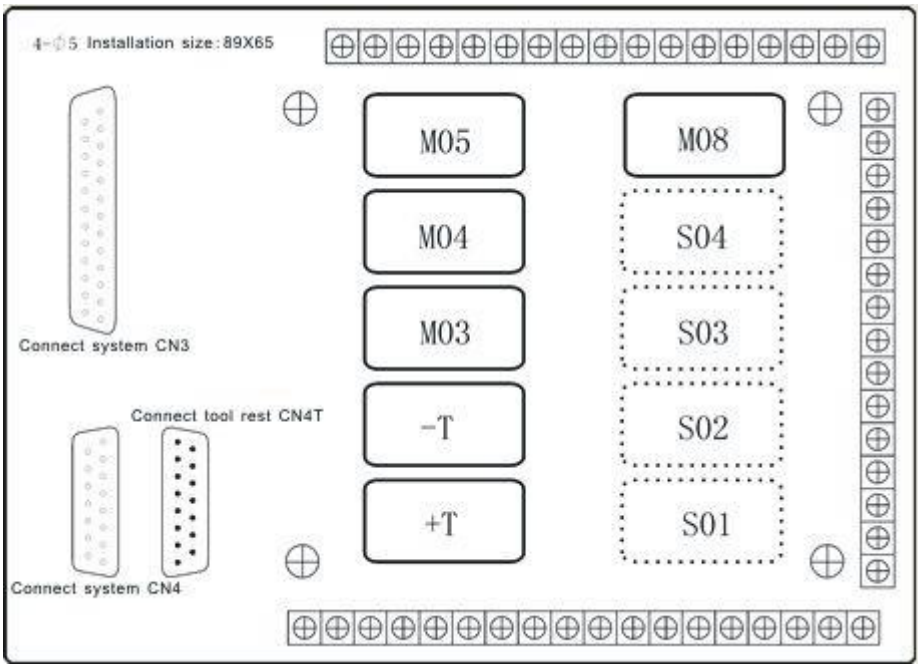
4.9 Electrical Interface Module: (Appliance plate of lathe)

Our company produces an electrical interface module for this lathe controller.

I/0-1 socket and CN3 are pin-for-compatible with CN3 of this system.

CN4 is pin-for-compatible with CN4 of this system.

CN4T can connect directly to an electric tool turret: 1-T1, 2-T2, 3-T3, 4-T4, 5-T5, 6-T6, 7-T7, 8-T8, 9-0V, 10-power supply +12V, 14-T0K.



4.9.1 Tool rest control:

+T, -T must be controlled by an external AC contactor. C3 is a common port of +T, -T and M08.

4.9.2 Spindle control:

C1 is the common port for M03 and M04.

C2 is the common port for M05. M5B is Normally closed.

4.9.3 Spindle gear control:

C4 is the common port for S1 and S2. S1B and S2B are Normally closed. C5 is the common port for S3 and S4. S3B and S4B are Normally closed.

Appendix A. Detailed parameter settings

NB – these explanations may not match the actual parameter numbers, or be complete but are here as a guide to help with some of the parameter settings.

Explanation of User Parameters

1, Cycle G71/G72 default feed thickness (10um)

It is set for every default infeed (radius) in X-axis direction when it isn't set in G71 code; unit: 0.01mm. It is set for every default infeed in Z-axis direction when it isn't set in G72 code; unit: 0.01mm.

2, Cycle G71/G72 default backward distance (10um)

It is set for every default backward move (radius) in X-axis direction when it isn't set in G71 code; unit: 0.01mm.

It is set for every default backward move in Z-axis direction when it isn't set in G72 code; unit: 0.01mm.

3, G71/G72/G173 instruction

It is for automatically smoothing when using G71/G72/G73 code. When set to 0, means no; when set to 1, means yes.

4, G71/G72/G73 default X remain (10um)

It is set for every default remain of smoothing (diameter) in X-axis direction when it isn't to be set in G71/G72/G73 code; unit: 0.01mm.

5, G71/G72/G73 default Z remain (10um)

It is set for every default remain of smoothing (diameter) in Z-axis direction. When it isn't set in G71/G72/G73 code; unit: 0.01mm.

6, G73 cutting times. It is for default cycle times when it isn't set in G73 code.

7, G73 X rough thickness (10um). It is for default rough thickness of X axis when it isn't set in G73 code.

8, G73 Z rough thickness (10um)

It is for default rough thickness of Z axis when it isn't set in G73 code.

9, G74/G75 default feed thickness (10um)

It is for every default infeed in Z-axis direction when it isn't set in G74 code; unit: 0.01mm.

It is for every default infeed (radius) in X-axis direction when it isn't to be set in G75 code; unit: 0.01mm.

10, G74/G75 default backward distance (10um)

It is for every default retract in Z-axis direction when it isn't set in G74 code; unit: 0.01mm.

It is for every default retract (radius) in X-axis direction when it isn't set in G75 code; unit: 0.01mm.

11, G76 finish turn times. It is for default cycle times when it isn't set in G76 code (times: 1-99)

12, G76 quit length (1/10 lead)

It is for default length of retract chamfer when it isn't set in G76 code. The length is 1/10 of thread lead.

13, G76 thread tooth angle (degree)

It is for default angle of thread tooth when it isn't set in G76 code. Unit: degree.

14, G76 minimal cutting depth (10um) [X axis radius]

It is to set minimal cutting depth (radius) of G76.Unit: 0.01mm

15, G76 finish turn remaining (10um) It is for default remaining of finish turn when it isn't set in G76 code. Unit: 0.01mm.

16, X program mode [**1** means Radius, **0** means Diameter]

There are two programming modes, when it set as 1 that means radius programming mode, when set as 0 means that diameter programming mode

17, Running program needs Spindle running [**1** means Yes, **0** means No]

It is for interlock between run program and run spindle, when set as 1 means that program won't run without running spindle; when set as 0 means that running program won't check if the spindle is running.

18, Set M20 the number of times of auto-running [Negative number means infinite loop]

It is for times of run M20 code in the program, negative number means run indefinitely.

19, Set part count. It is for display and set the time of run M20, it is also workpiece number.

20, G92 quit length (1/10 lead) It is to set default length of quit and retract, the length=thread lead * 0.1.

21, G01/G02/G03 line delay (ms) [>100] It is to set delay time between G01/G02/G03, it is to solve over-cutting in the corner.

22, G00 line delay (ms) [>100] It is to set delay time after run G00; it is effective when more than 100ms.

23, Hand wheel acceleration [50-100] it is to set hand wheel smooth acc/deceleration. The smaller it is, the faster the acc/deceleration is, but may cause vibration.

24, G76 Q [8 means thick pass forward number] it is to set the value of Q in G76, set as "8"; it is the times of feeding in roughing.

200, system screen protect times [>=2minutes]

201, G92/G76 delay time (ms) [>100] It is to set a delay time before check Z pulse when processing screw cutting.

Explanation of Speed Parameters:

1, X-axis's G00 speed (mm/min)

2, Z-axis's G00 speed (mm/min) Max 30000 (unit: mm/min)

Note: the value must take machine configuration into consideration, set wrongly could be damaging to the machine

3, Manual maximum feed speed (mm/min)

It is limitation of max feed speed in manual, Unit: mm/min. (reference speed=G00 speed * 0.5)

4, Auto Maximum feed speed (mm/min)

It is the max of feed speed in auto, Unit: mm/min. the speed could be faster than G00 speed.

5, G01/G02/G03 default speed (mm/min). Default speed of G01/G02/G03 when it isn't set in auto running. Max: 5000 (unit: mm/min)

6, Null run speed (mm/min) speed of null run simulation. (Press "simulate" to dry run) Max: 30000. (Unit: mm/min)

7, Feed axis`s manual speed (mm/min). It is the speed of feeding axis in Manual. Range: <max feeding speed (unit: mm/min) Note: in Manual, press "F", to refresh the parameters automatically.

8, Spindle`s manual speed (rpm). It is speed of spindle in manual. Unit: rpm.

Note: in Manual, press "S", to refresh the parameters automatically.

9, Beginning feed speed (mm/min) It is beginning speed of feeding axis when acc/deceleration. When it is smaller than acceleration/deceleration, accelerate/decelerate start from the beginning feed speed. When it is bigger than acceleration/deceleration, the speed is reached directly.

Note: it is related with machine configuration, in general, stepper systems are less than 100, and servo systems are less than 500.

10, Jump speed at continuous track (mm/min)

It is to increase the continuous movement when running multiaxial track-interpolation. Example: when it is 300, the speed of X axis (multiaxial track-interpolation) up from F800 to F1600, 800(=1600-800)>300, so the process is up from F800 to F1100, and then F1600.

11, Limit G1/G2/G3 axis speed [**1** means Yes, **0** means No]

It is to limit the max speed of each axis when G1/G2/G3 interpolating.

12, X G1G2G3 max speed (mm/min)

It is for the Max running speed of X-axis when G1/G2/G3 interpolating.

13, Z G1G2G3 max speed (mm/min)

It is for the Max running speed of Z-axis when set G1/G2/G3 interpolating.

14, X acceleration [1～99999]

It is the time constant of X-axis acc/deceleration, the bigger it is, the faster the acc/deceleration is.

Note: This value depends on the machine structure, the heavier the load is, and the smaller the value is. With stepper system, the value should less than 15000.

15, Z acceleration [1～99999] is the time constant of Z-axis acc/deceleration, the bigger it is, the faster acc/deceleration is.

Note: This value depends on the machine structure, the heavier the load is, and the smaller the value is. With stepper system, the value should less than 15000.

16, Auto run acceleration [1-500]

It is to set constant of acc/deceleration in auto. The range is from 1-500.It is

mainly to distinguish Auto and Manual, if the difference is too much, set it effectively.

17, Handwheel acceleration [500--32000]

It is to set the rate of acc/deceleration of Hand wheel. Range is from 500-32000.

18, Run program Handwheel acceleration [>500]

It is to set the rate of acc/deceleration of Hand wheel when running program. Range is from 500-32000.when the value is less than 500, it is invalid.

19, Run program Handwheel G00 speed (mm/min) [>10]

It is the G00 speed when triggered by Handwheel in testing machine. Invalid when <10.

20, Handwheel X limit speed (mm/min)

It is to limit the handwheel max speed of X-axis when using handwheel in manual. Note: it is valid when >100, otherwise invalid.

21, Handwheel Z limit speed (mm/min)

It is to limit the handwheel max speed of Z-axis when using handwheel in manual. Note: it is valid when >100, otherwise invalid.

22, Make thread Z acceleration [Servo motor is 0]

It is to set the rate of Z-axis acc/deceleration, the range is from 300 to 90000.

Note: with stepper system, the smaller it is, the safer is. When <300, invalid.

23, Make thread X acceleration [Servo motor is 0]

It is to set the rate of Z-axis acc/deceleration, the range is from 300 to 90000.

Note: with stepper system, the smaller, the safer. When <300, invalid.

24, Servo motor screw thread X axis Back speed

It is the speed rate of X axis in servo system when back in processing screw thread. Unit: 0.1 times.

25, Step motor screw thread X axis Back rise speed it is the start speed of X axis in step system when back in processing screw thread. Unit: mm/min.

Note: for safety, it should less than 100mm/min.

26, Step motor screw thread X axis Back Max speed

It is the Max speed of X axis in step system when back in processing screw thread. Unit: mm/min. Note: for safety, it should less than 100mm/min.

27, Acceleration type [0 means line, 8 means curve] Set type of acc/deceleration. Set 0 means line type, set 8 means curve type.

Note: In normal condition, set line type in step system. Set curve type in servo system.

28, Curve initial acceleration [>=10] Initial acc/deceleration constant when set curve type. It is valid when >=10.

29, Curve acceleration [>=10] It is second ace/deceleration constant when set curve type. It is valid when >=10.

30, Curve max acceleration [>=500] Max acc/deceleration constant when set curve type. It is valid when >=500, otherwise the acc/deceleration constant is with line type of each axis.

31, X go home speed (mm/min) Speed of X-axis when go home in forward

direction. Unit: mm/min. Range is less than the G00 speed of X-axis.

32, X go home reverse speed (mm/min) Speed of X-axis when go home in reverse direction. Unit: mm/min. Range is 20-500.

Note: To ensure accuracy, the smaller it is, the higher the accuracy is. Once optimally set, don't change.

33, Z go home speed (mm/min) Speed of Z-axis when go home in forward direction. Unit: mm/min. Range is less than the G00 speed of Z-axis.

34, Z go home reverse speed (mm/min) Speed of Z-axis when go home in reverse direction. Unit: mm/min. Range is 20-500.

Note: To ensure accuracy, the smaller it is, the higher the accuracy is. Once optimally set, don't change.

35, G96 spindle min speed (rpm). Minimum spindle speed running G96 code.

36, Spindle first max speed (rpm) first max speed of spindle, it is also the speed when voltage is 10V. Unit: r/min

37, Spindle second max speed (rpm) it is the second max speed of spindle, it is also the speed when voltage is 10V.unit: r/min

38, Spindle third max speed (rpm)

It is the third max speed of spindle; it is also the speed when voltage is 10V. Unit: r/min

39, Spindle fourth max speed (rpm)

It is the fourth max speed of spindle; it is also the speed when voltage is 10V. Unit: r/min

40, Second Spindle max speed (rpm)

It is the max speed of the second spindle; it is also the speed when voltage is 10V.unit: r/min

41, Reverse compensation mode (**0** means A; **8** means B) It is reverse compensation mode of arc gap. 0 means A mode.(A mode is the bigger it is ,the faster the speed is. the speed should not be bigger than 1000mm/min, the speed also is related with the value of reverse gap compensation 8 means B mode. (B mode - the speed depends on the related parameters.)

42, mode B reverse compensation speed (mm/min) it is the speed of reverse compensation in B mode. Unit: mm/min.

42-1, mode B reverse compensation Beginning feed speed(mm/min) It is beginning speed of reverse compensation in B mode.it is valid when it >10. Unit: mm/min.

, mode B reverse compensation acceleration (mm/min/s)

It is the constant of reverse compensation acceleration It is valid when it >=10. Unit: mm/min

43, Handwheel stop speed (mm/min) [>100] the speed when handwheel stop. The bigger it is, the shorter the stop time is.

58, Forced limit drop speed critical (mm/min)

It is starting drop speed when it is forced limit. When servo system, it is 1.

Explanation of Axis parameters:

1, Feed axis band switch [**1** means Yes, **0** means No]

It is to alter feeding axis's rate. 1 means selection external band switch (it is the band switch in additional panel.) to alter.0 means use Feed rate+/- on inbuilt panel.

2, Spindle band switch [**1** means Yes, **0** means No]

It is to alter spindle axis's rate.1 means selection external band switch (it is the band switch in additional panel.) to alter.0 means use SP rate+/- on inbuilt panel.

3, X-axis`s negative scope (mm) It is the coordinate value of X-axis soft limit in maximum negative direction.

4, X-axis`s positive scope (mm) It is the coordinate value of X-axis soft limit in maximum positive direction.

5, Z-axis`s negative scope (mm) It is the coordinate value of Z-axis soft limit in maximum negative direction.

6, Z-axis`s positive scope (mm) It is the coordinate value of Z-axis soft limit in maximum positive direction.

7, Spindle stop time (10ms) Braking time of spindle, the shorter it is, the faster the brake is.

8, Spindle stop long signal [**0** means No, **1** means Yes] When it is 1, the signal of spindle is a long signal, when it is 0, it is short signal.

9, Check Spindle encoder: [**1** means Yes, **0** means No] It is for whether the system checks the signal of spindle encoder, also the spindle's position feedback.

1 means check, 0 means no check. Spindle gear ration must be 1:1 with spindle encoder.

10, Spindle encoder pulse is the feedback pulses of each rev of spindle encoder. It is lines (of SP-encoder) * 4.

11, Soft limit invalid [D2X; D3Y; D4Z; D5A; 0 invalid, 1 valid]

It is to set each axis has a soft limit. Every axis is set individually. D2:X; D3:Y(C); D4:Z, D5:A; D6:B.

Example: Soft limit of X-axis is valid, the bit parameter is 00000100.

12, X-axis`s reverse compensation (um)

It is the value of reverse compensation (radius), when X-axis is running in negative direction. When X-axis runs in negative direction, system compensates with the value.

13, Z-axis`s reverse compensation (um)

It is the value of reverse compensation (radius), when Z-axis is running in negative direction. When Z-axis runs in negative direction, system compensates with the value.

14, X-axis's direction signal [**1** means normal, **0** means reverse]

It is to change the direction of X-axis. When it is 0, the direction of code is

opposite to the direction of moving. When it is1, the direction is same.

15, Z-axis's direction signal [**1** means normal, **0** means reverse]

It is for change the direction of X-axis. When it is 0, the direction of code is opposite to the direction of moving. When it is1, the direction is same.

16, Close feed electronic gear [**1** means Yes, **0** means No]

It is to close the electronic gear of the feeding axis.1 means close, 0 means no close.

17, X-axis's electronic gear numerator (1-999999)

Is the numerator of X-axis electronic gear, Multiplication ratio of axis X's instruction (X_CMR)

18, X-axis's electronic gear denominator (1-999999)

Is the denominator of X-axis's electronic gear, Frequency-division coefficient of axis X's instruction (X_CMD)

19, Z-axis's electronic gear numerator (1-999999)

Is the numerator of Z-axis's electronic gear, Multiplication ratio of axis Z's instruction (Z_CMR)

20, Z-axis's electronic gear denominator (1-999999)

Is the denominator of Z-axis's electronic gear, Frequency-division coefficient of axis Z's instruction (Z_CMD)

P17-P20 parameters: Effective Range: 1-999999 Unit: Non User: Upon operating administrators Initialization: 1 Effective time: Immediate Explain: When lead screws with different screw pitches are configured with motors of various step angles, or with servo motors of different pulse number per round, or connections are realized through different gearing, the programmed values can remain consistent with the actual moved distance by setting the parameter of the electronic gear ration of the system.

CMR/CMD = P/ (L * 1000)

CMR: Numerator of gear ratio CMD: Denominator of gear ratio P: pulse number per motor round

L: Moved distance per motor round (mm)

The value of CMD/CMR is the pulse equivalent, which tells the moved distance per pulse, with its unit as 0.001mm.

Example1:

The motor rotates one revolution every 5000 pulses, after which the machine tool moves 5mm, then:

CMR/CMD = 5000 / (5 * 1000) = 1/ 1

That is to say, we can set the values as: CMR=1, CMD=1. Here, the pulse equivalent is 0.001mm.

Example2:

The motor rotates one revolution every 5000 pulses, after which the machine tool moves 10mm.

CMR/CMD = 5000 / (10 * 1000) = 1 / 2

That is to say, we can set the values as: CMR=1, CMD=2. Here, the pulse equivalent is 0.002mm.

Basically

The numerator is the number of pulses per rev of the stepper or servo motor.

The denominator is the distance the axis moves in um.

21, XZ positive limit [**0** open, **1** close]

Is the type of limit switch in positive direction. 0 means the switch is normally open, 1 means it is normally closed.

22, XZ negative limit [**0** open, **1** close]

Is the type of limit switch in negative direction.0 means the switch is normally open, 1 means it is normally closed.

23, float zero bit parameter [D3X; D4(C)Y; D5Z; D6A; [**0** machine Zero; **1** float Zero]

It is for whether the floating zero function is valid, every axis is set individually. Bit parameter. D3:X; D4:C(Y); D5:Z; D6:A; D7:B.1 means the axis has a floating zero point,0 means use machine zero point.

Example: X set float zero point, the bit parameter is 00001000.

24, X coordinate float zero set. Is the coordinate value of X-axis float zero point.

25, Z coordinate float zero set. Is the coordinate value of Z-axis float zero point.

26, Feed axis home [1 means Not in use, 0 means prompt, 8 compulsory , 9 must comply] It is the request that feeding axis go home. There is four kinds way of go home as follow:

1 means no request. When boot every time, no prompt and no limitation; 0 means prompt, when boot every time, there will be a prompted screen;

8 means enforcement, when boot every time, there will a prompted screen, and then, if the system don't go home, it will note "feed axis don't go home", and won't run the program. 9 means enforced, when boot every time, there will a prompted screen, and then, if the system doesn't go home, it will note "feed axis don't go home" and feed axis won't move.

27, Feed axis home mode [**0** reverse check, **1** reverse No check, **2** No reverse check, **3** No reverse No check]

It is the mode that checks the switch and Z pulse of motor's encoder when feeding axis go home:

When it is 0, go home, after hitting the switch, moves in reverse direction until the switch is disengaged, and then check the Z pulse of encoder.

When it is 1, go home, after hitting the switch, move in reverse direction until the switch is disengaged.

When it is 2, go home, after hitting the switch, move forward until the switch is disengaged, and then check the Z pulse of encoder.

When it is the rest, go home, after hitting the switch, move forward until the switch is disengaged.

28, Home reverse direction [D2X; D3C(Y); D4Z; D5A; D6B; D8Zpriority]

It is for the direction and priority for feeding axis to go home. It is a bit parameter, each axis is set individually. D2:X; D3:C(Y); D4:Z; D5:A; D6:B; D8:XZ priority,1 means negative direction, 0 means positive, D8 controls the priority that X&Z-axis go home.1 means Z-axis first, 0 means X-axis first.

Example: when X-axis is set to go home in negative direction with Z axis homing first, the bit parameter is 100000100.

29, Home NC switch bit set [D0X; D1C(Y); D2Z; D3A; D4B; D7: Manual/Auto cut automatically ;1: Close ; 0: Open]

It is the mode of home switch, set alone, it is bit parameter. D0:X; D1C(Y); D2Z; D3A; D4B; D7:Manual/Auto cut automatically, when after program in auto enter manual condition automatically ;1: NC(normal close); 0: open.

Example: If X&Z axis are NC switch, the bit parameter is 000000101.

30, X check zero max length (unit:100um)

It is the length to check zero pulse of encoder when go home and after disengaged switch.

Note: the value must be less than the length of one rev!

31, Z check zero max length (unit:100um)

It is the length to check zero pulse of encoder when go home and after disengaged switch.

Note: the value must be less than the length of one rev!

32, X Home offset (unit:10um,-9999~+9999)

It is the offset that X-axis moves in G00 speed when go home, after check zero pulse.unit:0.01m.

33, Z Home offset (unit:10um,-9999~+9999)

It is the offset that Z-axis move in G00 speed when go home, after check zero pulse.unit:0.01m.

50, Have Spindle class control : [**1** means open,**0** means close]

It is whether the spindle is rotating when changing gears.1 means spindle is open, 0 means close.

51, Spindle class speed (1/100rpm)

It is the speed that the spindle rotates when changing gears.

52, Spindle class direction: [0 means M03,1 means M04]

It is the direction that the spindle rotates when changing gears,1 means reverse,0 means forward.

53, Spindle class stop time (unit:10ms)

It is the time that the spindle stops (M05) when changing gears.

54, Spindle class time (unit:10ms)

It is the time that the spindle runs in low speed. (unit:10ms)

55, Spindle stop time (unit:10ms)

It is the delay time between cancel M03/M04 and boot M05. (unit:10ms)

80, XZ axis coordinate plan [D2 Z workpiece, D3 X workpiece, D4 Z tool, D5 X tool, D6 Z circumrotate , D7 X circumrotate]

It is bit parameter, D2:Z axis in workpiece coordinate system; D3:X axis in workpiece coordinate system; D4 is Z axis in machine coordinate system; D5 is X axis in machine coordinate system. D6 is Z axis whether is rotation axis; D7 is X axis whether is rotation axis.1 means valid/yes; 0 means invalid/no.

404, SP motor direction (**0** reverse, **1** normal)

It is the direction of spindle motor, **0** reverse, **1** normal.

405, SP-axis's electronic gear (**0** Yes, **1** No)

It is for whether the spindle uses electronic gearing.

406, SP-axis's electronic low gear numerator (1-999999)

It is the numerator of SP-axis's electronic low gear in low gear.

407, SP-axis's electronic low gear denominator (1-999999)

It is the denominator of SP-axis's electronic low gear in low gear.

408, SP-axis's electronic high gear numerator (1-999999)

It is the numerator of SP-axis's electronic low gear in high gear.

409, SP-axis's electronic high gear denominator (1-999999)

It is the denominator of SP-axis's electronic low gear in high gear.

410, Interpolation tap SP name [91 X,92 Y/C,93 Z,94 A,95 B]

It is the axis to be used when tapping.

411, Interpolation tap mode [**2** follow encoder; **3** interpolation to SP]

It is the control mode of tapping.

412, SP tooth number (<P413)

Is number of teeth on spindle pulley <=P413.

413, Encoder number (>P412)

Is number of teeth on SP-encoder pulley >=P412.

Explanation of Tool parameters

1, Active tool function: [**1** means Yes, **0** means No] It is whether to activate electric turret (tool).

Note: when the machine doesn't have an electric turret, the parameter be set as 0, and also use Txxxx code to control tools and radius/tool compensation.

2, Active tool numbers in electric turret. It is the total number of electric turret.

Example: when lath machine have four-electric turret and four-linear turret, in "Redeem" screen, press F7 "Set", input tool total count is 8,and the parameter is 4,so T1-T4 means in electric turret,T5-T8 means in linear turret.

3, Lathe type

It is type of lathe machine's structure. 0: turret in front of horizontal lathe;

1: turret behind of horizontal lathe; 2: turret in front of vertical lathe; 3: turret behind vertical lathe.

4, Tool positive rotate max-time(s)

It is the max-time that the turret is changing tools automatically. When over the time, the system stops ATC and alarms.

5, Delay time after tool positive rotate (ms)

It is the delay time to check the tool signal (Tok) after the turret rotated

6, Delay time after tool stops (ms)

It is the delay time that is between the turret forward rotate is okay and stop, also is between cancel forward rotate signal (+T) and output reverse rotate signal (-T).unit: Ms.

7, Tighten time of tool reverse rotate (ms)

It is the time to tighten tool in reverse rotate, also is the time that output the signal of "-T".

Note: the bigger the value is, the hotter the motor is.

9, Have signal TOK (1 means have)

It is for whether check the signal of "TOK", 1 means check it, 0 means not.

10, C Tool radius compensation's establish [**0** means A, **1** means B]

11, C Tool radius compensation's cancel [**0** means A, **1** means B]

20, Active tool mode: [**1** means normal, **0** means coding tool]

When set as coding tool, must restore in PLC of coding tool, detailed code and edit ATC program are press "F1"in Diagnosis condition.it needs to input password.

Explanation of Other parameters

1, Set sub-panel type: [**1** hand held, **0** panel]

It is the type of handwheel, 1 means hand held type, 0 means handwheel in panel(C panel). Note: when the parameter is 1(P1=1), CN11(handwheel port) couldn't be used to alter axis, so P1,P2 only set as 0.A/X/Y(C)/Z to select axis, X1/X10/X100/OFF to select increment.

2, Lathe outside chuck : [**1** extroversion, **0** diffidence] ?

3, Use control switch : [**1** Yes, **0** No]

It is for whether the tamper switch is valid.1 means valid, 0 means invalid.

4, Have auto lubrication (**0** Yes/ **1** No)

Is whether the auto lubricate is valid. 1 means valid, 0 means invalid. Note: auto lubricate is decided by working time.

5, Auto lubricate time (0.01s) It is the duration of auto lubricate, also time that M32 is valid.unit:0.01s.

6, Auto lubricate stop time (s) It is the interval between lubrication cycles, also the interval that twice M32 is valid: unit: s.

7, Door switch checking M12 (**0** no, **1** yes) It is whether the system checks the signal of safe-door. 0 means there isn't safe-door, 1 means there is safe-door and check it.

Note: for the check of safe-door, it is realized by M12.

When set valid and M12 is also valid, in Manual system could work, but not in Auto.

8, Door switch (Model: **0** open, **1** close) It is type of safe-door switch. 0 means

normally open type NO, 1 means NC(normal close).

9, Bit parameter

D0：Null;

D1 ： “1” Start CNC system, clear part Number.;

D2：“1” Automatic space before letter when edit program;

D3：Null;

D4：Null;

D5：“1” Do not stopping SP and cooling when pressing “Reset”;

D6：“1” G00 XZ’ speed by oneself;

D7：“1” Tool’ redeem by oneself;

D8：“1” Save SP chuck (M10/M11) state when power off;

D9：Tool redeem input Mode1 or Mode2;

D10：“1” Program edit automatic compositor Line;

D11：“1” First SP +10V output from second output port;

D12：“1” Shield skip function (“/”is invalidation);

D13：“1” Shield go home function;

D14：“1” Shield “run” key;

D15：“1” Tool redeem display relative, “0”absolute; 10, Auto count part : [**1** means Yes, **0** means No]

It is for whether to count workpiece automatically, 1 means count, 0 means not.

11, Program edit number increase

12, Inner parameter

13, Does lock for both Spindle & chuck (**0** means no)

It is for whether interlock between running spindle and chuck.1 means interlock, 0 means no.

14, Is available keys of lubricate & coolant as running (**0** means no)

It is for whether the coolant is valid in Auto. When P14=0, it is invalid.1 means valid.

15, Chuck clamp M10/loose M11 checking (1 means need)

16, Tailstock forward M79/backward M78 checking (1 means need)

17, Servo ALM (**0** open, **1** close) Check ALM of servo drive (Pin12 in CN5),1 means normal close,0 open.

18, SP ALM1 (**0** open, **1** close) Check ALM1 of Spindle (Pin5 in CN3),1 means normal close,0 open.

19, Tool ALM2 (**0** open, **1** close) Check ALM2 of tool (Pin2 in CN10),1 means normal close,0 open.

20, Chuck control signal (**0** single, **1** doubleM10/M71)

It is to set the chuck control signal is single or double, M10/M71 are the two

signal.

When set as 0, only use M10 to control chuck. When M10 is valid, tighten chuck, otherwise invalid.

When set as 1, use M10&M71 to control chuck. When M10 is valid and M71 is invalid, tighten chuck, otherwise when M10 is invalid and M71 is valid, loosen chuck. (M10 output M10, M11 output M71)

Check whether it is in place, P40 in Axis parameter, 1 means check, 0 means no. When set as check, M10 is in place, and the system also checks whether M40 is valid automatically, when M40 is valid, do next step. Loosen chuck M11, chuck is in place, and the system also check whether M12 is valid automatically, when M12 is valid, do next step.

21, Tailstock control signal (**0** single, **1** double M79/M73)

22, Outside chuck control (**0** no, **1** yes)

It is for whether use external switch (button) to control tighten/loosen, it is reciprocating signal, one time is valid, tighten, next time is invalid, loosen. When set 0 means without external switch, when with external switch, the signal is M16.

23, Outside Tailstock control (**0** no, **1** yes)

24, M10, M11 short signal time(s)

It is for stay time that output M10, M71 is short signal.0 means them is long signal. (unit: s)

25, M79, M78 short signal time(s)

26, Emergency Stop (**0** open, **1** close). Is the type of emergency stop1-switch in operation panel. For safety, advice set 1--Normally closed.

27, Emergency Stop2 (**0** open, **1** close)

Is the type of emergency stop2-switch in handwheel or external switch. (Pin5 in CN11). For safety, advice set 1--Normally closed.

28, Run status output M69 STOP output M65 (**0** invalid, **1** valid)

29, Alarm status output M67 (**0** invalid, **1** valid)

30, Set language (**1** Chinese, **0** English)

31, Enable I/O PLC program

32, Enable High speed I/O PLC program

33, HY make run signal: [**1** means Yes, **0** means No]

34, HA make stop signal: [**1** means Yes, **0** means No]

35, Soft-limit without home as manual: [**1** Yes, **0** No]

36, Set system time: [year-month-day-hour-minute]

37, Velocity of RS232

[0=7200；1=9600；2=14400；3=19200；4=38400；5=57600；6=115200]

Note: the bigger the value is, the more unstable it is. The bit rate of both ends must match

38, Lock Manual rapid function key: [**8** Yes]

39, Special parameter

40, Special parameter

41, Bake current parameter (Save all parameters overwriting any defaults)

It is set to ex-factory values. Used to save current parameters after testing system thoroughly. ADVICE – DON'T – backup to U disk

42, Resume original parameter (can't if baked)

601, Make system Step Motor

602, Make system Servo Motor

Explanation of Coordinate Paramers

These parameters coordinate multiple functions, namely a six workpiece coordinate system and a machine coordinate system G53.

A machining program can set a single workpiece coordinate system and can also set up multiple workpiece coordinate systems. The workpiece coordinate system can be changed to move its origin. That is the value of the parameter in the coordinates of its own coordinate origin (zero) coordinate value in the machine coordinate system.

Six workpiece coordinate systems are available G54 to G59, the coordinate system settings interface can be used to modify 6 origins of the workpiece coordinate system coordinate value in the machine coordinate system.

1, X of work coordinates G54 (mm)

2, C(Y) of work coordinates G54 (mm)

3, Z of work coordinates G54 (mm)

4, A of work coordinates G54 (mm)

6, X of work coordinates G55 (mm)

7, C(Y) of work coordinates G55 (mm)

8, Z of work coordinates G55 (mm)

9, A of work coordinates G55 (mm)

11, X of work coordinates G56 (mm)

12, C(Y) of work coordinates G56 (mm)

13, Z of work coordinates G56 (mm)

14, A of work coordinates G56 (mm)

16, X of work coordinates G57 (mm)

17, C(Y) of work coordinates G57 (mm)

18, Z of work coordinates G57 (mm)

19, A of work coordinates G57 (mm)

21, X of work coordinates G58 (mm)

22, C(Y) of work coordinates G58 (mm)

23, Z of work coordinates G58 (mm)

24, A of work coordinates G58 (mm)

26, X of work coordinates G59 (mm)

27, C(Y) of work coordinates G59 (mm)

28, Z of work coordinates G59 (mm)

29-1, How to set up the workpiece coordinate system?

a).press “F8-coordinate”key,select related workpiece coordinate system(54-59)

b).move machine to suitable position that easy to measure in manual, measure the related coordinate value between this point (zero point in the workpiece) to Home of G53.

c).press “F7-set coordinate”, press “X” key and enter, insert the value to dialog, and enter.

d).press “F7-set coordinate”, press “Z” key and enter, insert the value to dialog, and enter.

It is finished now, enter coordinate screen in parameter, it could be seen the parameter values have been set well, it is offset value between workpiece coordinate and machine coordinate.

29-2, How to adjust the offset value?

If workpiece coordinate system is set well, when it is needed to adjust the offset value, it could be set by entering the coordinate parameter screen.

Steps follow:

In the coordinate parameter screen, selected the parameter, press “Enter”, and pop up dialog, input the offset value (also Increments, example: offset 10mm in negative direction, input -10, it is okay), press “Enter”.

Explanation:1 When the parameter is altered correctly, the coordinate main screen will refresh the corresponding coordinate value.

Brackets in these parameters, means the sum, which is offset or adjusted every time.