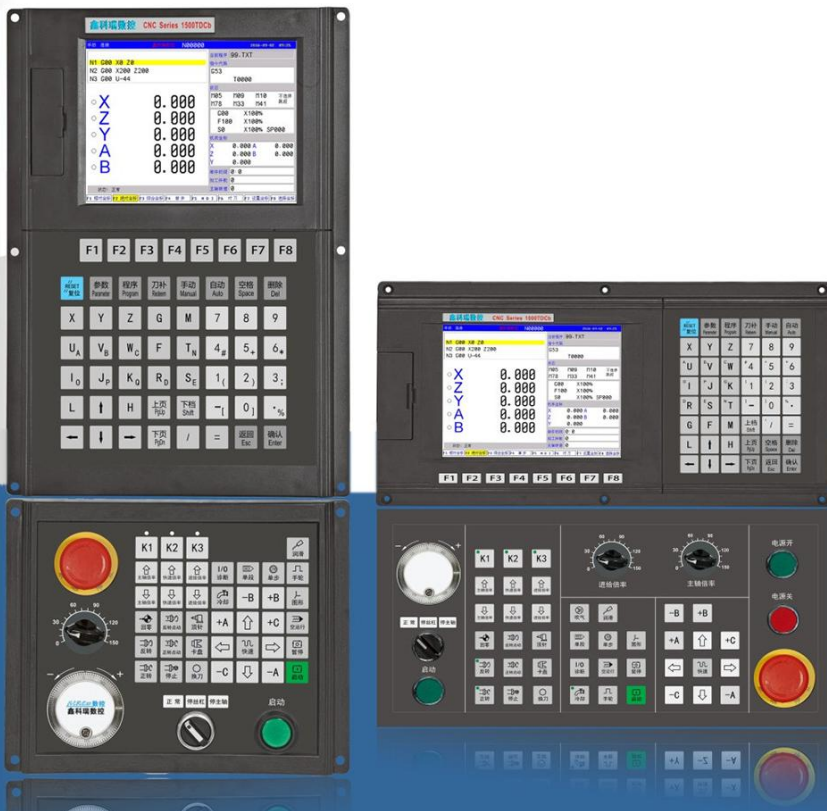


NEWKer



Making the most ideal and practical product to service CNC world

NEW1000/1500TDC Controller

Operation Manual

CHENGDU NEWKer CNC-TECHNOLOGY CO.,LTD

CONTENTS

Chapter 1	Preface	1
Chapter 2	Controller Connection and Installation	3
2.1	System electrical specification	3
2.2	System technical index	3
2.3	Environment of operation	3
2.4	System installation and connection	4
2.5	System installation dimension	4
2.6	System rear view	7
2.7	Interface connection graph	7
2.7.1	Electric turret CN4 female socket(DB15)	7
2.7.2	Spindle Encoder CN9 Male socket	9
2.7.3	Communication CN6 female socket	10
2.7.4	I/O 1 signal connection CN3 female socket(DB25)	11
2.7.5	I/O 2 signal connection CN10 female socket(DB25)	12
2.7.6	I/O 3 signal connection CN16 female socket(DB25)	13
2.7.7	X/Y/Z Driver control signal connection CN5 male socket(DB25)	13
2.7.8	A/B Driver control signal connection CN6 female socket(DB15)	14
2.7.9	External switch, electrical hand wheel CN11 male socket (DB15)	15
2.7.10	Axis position feedback signal CN13 male socket(DB26)	18
2.7.11	Absolute type controller setting steps	18
2.8	Machine electric element installation	19
2.8.1	Limit position	19
2.8.2	Lathe home point	21
2.8.3	Switch signal control	23
2.8.4	System alarm signal	25
2.8.5	User-defined output signal	26
2.9	Relay board of controller(optional)	27
2.9.1	Tool turret control	28
2.9.2	Spindle control	28
2.9.3	Spindle gear control	28
Chapter 3	Operation	29
3.1	Summary	29
3.2	Operation panel and switches	29
3.3	Keyboard instruction	29
3.3.1	Rate increase	29
3.3.2	Switches	29

3.3.3 Other keys.....	30
3.4 Manual operation.....	31
3.4.1 The key of manual operation.....	32
3.4.2 Manual continuous.....	34
3.4.3 Manual increment.....	34
3.4.4 Back to lathe home point (reference point).....	35
3.4.5 MPG(Hand wheel).....	36
3.4.6 Tool set.....	37
3.5 Auto operation.....	38
3.5.1 Automatic process.....	39
3.5.2 Start from any program line or tool number line.....	39
3.5.3 Start program.....	40
3.5.4 Stop processing program.....	40
3.5.5 Real-time control in automatic mode.....	40
3.5.6 MDI Mode.....	41
3.5.7 MPG(hand wheel) mode.....	41
3.5.8 DNC Function.....	42
3.6 Safe operation and Alarm.....	42
3.6.1 Emergency stop.....	42
3.6.2 Reset system.....	43
3.6.3 Alarm.....	43
3.7 Parameter operation.....	43
3.7.1 User parameter (processing).....	44
3.7.2 Speed parameter.....	48
3.7.3 Axis parameter.....	52
3.7.4 Tool parameter.....	59
3.7.5 Other parameter.....	60
3.7.6 Coordinate.....	68
3.7.7 Password.....	72
3.8 Set tool compensation.....	72
3.8.1 Radius compensation.....	72
3.8.2 Length compensation.....	73
3.8.3 Posit tool.....	73
3.8.4 Set quantity.....	76
3.9 Screw compensation.....	76
3.10 Diagnosis.....	80
3.11 Operation of program.....	82
3.11.1 Editing.....	83
3.11.2 Copy.....	85
3.11.3 Delete.....	85
3.11.4 Rename.....	85
3.11.5 Information.....	85
3.11.6 Checking program.....	85
3.11.7 Folder management.....	85

3.11.8 Select automatic to run program.....	85
3.11.9 Program communication.....	86
3.11.10 U-disk management.....	86
3.11.11 Convert offline file into G code program.....	87
Chapter 4 Programming.....	89
4.1 Program coordinate.....	89
4.2 Program structure.....	89
4.3 Program instruction.....	90
4.3.1 Functional meaning of address symbol, data list.....	90
4.3.2 Program instruction table.....	91
4.4 Programming instruction and usage.....	98
4.4.1 Programming stipulation.....	98
4.4.2 Instructions.....	98
(1) Rapid location(G00).....	98
(2) Line interpolation(G01).....	99
(3) Arc interpolation(G02/G03).....	100
(4) Screw thread (G32)/Polygonal cutting(G809).....	102
(5) Circularity screw thread(G332、 G333).....	105
(6) Arc in three-dimensional space(G06).....	106
(7) Delay Instruction(G04).....	106
(8) Return Reference(G28).....	107
(9) Setup work piece coordinate system, max spindle speed of constant linear speed (G50).....	108
(10) Column or taper loop(G90).....	108
(11) End face loop(G94).....	111
(12) Screw thread loop (G92).....	113
(13) Usage of fixed cycle.....	118
(14) Fixed cycle of tapping (G93).....	119
(15) Column rough cutting loop (G71).....	120
(16) End face thick loop(G72).....	122
(17) Close cutting cycle(G73).....	123
(18) Finishing machining cycle (G70).....	124
(19) End face deep hole loop(G74).....	127
(20) Slot cutting loop(G75).....	128
(21) Complex screw thread cycle(G76).....	129
(22) Program loop(G22、 G800).....	133
(23) Part coordinate setup(G52), Setup or offset tool coordinate (G184、 G185)	133
(24) Back to starting point(G26、 G261-G268).....	135
(25) Back to G25 point(G25、 G61、 G611-G618).....	136
(26) Continue feed cutting(G60/G64).....	136
(27) Constant speed cutting(G96/G97).....	136
(28) Feed mode(G98、 G99).....	137

(29) Check skip(G31、 G311).....	137
(30) Work coordinate(G53/G54/G55/G56/G57/G58/G59).....	138
(31) Tool radius compensate G40、 G41、 G42.....	138
(32) Radius compensation of tool C.....	140
(33) Automatic chamfer I and fillet R:.....	144
(34) Polar coordinate program(G15/G16).....	145
(35) Polar coordinate interpolation(G12.1/G13.1).....	146
(36) Metric system and British system (G20/G21), absolute program and incremental program(G990/G991).....	148
(37) M instruction.....	149
(38) Chips breaking/shaping machine specific instruction(M114/M115)..	153
(39) Call program (M97/M98/M99).....	153
(40) Conditional wait or skip.....	154
(41) T tool function.....	155
(42) S、 SS Spindle speed.....	156
(43) F feed speed.....	156
(44) Macro program instruction(G65, G66, G67).....	156
(45) User-defined macro instruction(G120-G160,M880-M889).....	161
(46) Composite function of turning and milling.....	162
4.5 Program Sample.....	163
4.6 Usage for grinder.....	164
4.7 Instruction of following axis.....	165

Chapter 1 Preface

This system is a universal lathe CNC system developed by our company. The control circuit is using the latest industrial high-speed ARM processor, large-scale field and programmable FPGA technology, multi-layer PCB, the machine adopts the high integrated chip and surface mount components, the structure is more compact and reasonable so that make sure the reliability and stability of the system.

Real-time high speed control(the max rapid speed is up to 240 m / min, the max feed interpolation speed can reach 30 m / min), high accuracy; 800x600 dot matrix TFT LCD adaptive brightness LCD display, LED backlight brightness uniformity and long service life, and overcomes the display brightness by environmental temperature changes the shortcoming. Full English display, simple and convenient operation.

This system is based on the lathe of the two or three, four axis interpolation, closed-loop control and universal fully digital control system, powerful function and complete instructions, programming code accords with ISO international code standard. Directly control AC servo driver, if choosing the dual channel AC servo driver, it will higher cost-effective.

This manual details the programming and using method of lathe system.

Important notice:

1. When using this system for the first time, please read carefully all the details of each chapter so as to make it work more efficiently.
2. All the functions of the third and fourth axis are effective when configuring the designated system.
3. All the function of absolute-value must choose the absolute-value system, all the bus function must choose the bus system can be effective.
4. The “Run” button on the panel of system can be used when debugging (No.9-D14 parameter in other parameter to set “Effective” “Invalid”), user should install an external “Run” button when fitting system, otherwise may cause accident because of button life!!! So the system

prohibits using the button always, otherwise the any accident
consequences has nothing to do with us!!!

Chapter 2 Controller Connection and Installation

2.1 System electrical specification

- 32bits high performance industrial grade ARM+DSP+FPGA
- 128M User's storage space
- 800x600 TFT LCD adaptive brightness, LED backlight LCD
- Touch type key board with excellent operational sensitivity.
- RS232 communication port
- USB port
- Photoelectric encoder
- 4 and 8 position electrical turret or row turret.
- Highly anti-interference switching power supply.
- Two-way spindle of variable frequency and speed governing
- Manual pulse generator
- Band switch for feeding speed and spindle speed adjusting

2.2 System technical index

- Controllable axis: XYZAB 5 axis
- Linkage axis: XYZAB 5 axis
- Pulse equivalency: X axis with 0.001mm, Z axis with 0.001mm
- Maximum speed: 240m/min
- Processing speed: 0.01-30000mm/min
- Processing screw pitch: 0.1-1000mm in the metric system, 1-99 tooth/inch
- Minimal input unit: 0.001mm
- Programme size range: ± 99999.999 mm
- Programme coordinate system definition: IOS-841 international standard
- Programme code: IOS-840 international standard
- Mean Time Between Failure (MTBF): Greater than 6000 hours

2.3 Environment of operation

- Power supply: AC 220V (+10%,-15%), frequency 50Hz $\pm 1\%$
- Power source ≤ 100 W
- Running temperature: 5~45°C, relative humidity: 40~80%
- Storage and transportation temperature: 0~55°C, relative humidity less than 90%(40°C)

2.4 System installation and connection

At first, users should check whether the hardware is complete and compatible, such as: cnc system, driving power, servo motor, photoelectric encoder, electric tool carrier.

The installation of cnc system must be fastened tightly, with some spaces around to ensure the ventilation of air. Panel should be put in a place where it is not only convenient to operate and but also able to avoid hurt of heating by scrap iron.

Intense current, weak current must be put separately; cnc system and driver should be possibly away from the machine intense current. In order to reduce interference, all signal cables should be kept away from AC contactor. Photoelectric encoder, limit, basic point signal are advisably not to be connected directly to cnc system through intense current box. All power cords must be grounding.

Fix all plugs with screw. Forbid to insert and extract all cables when power is on.

During installation of cnc system, panel should avoid hurting by hard and sharp materials. If the painting of other part of machine is needed, please take off cnc system to keep it clean.

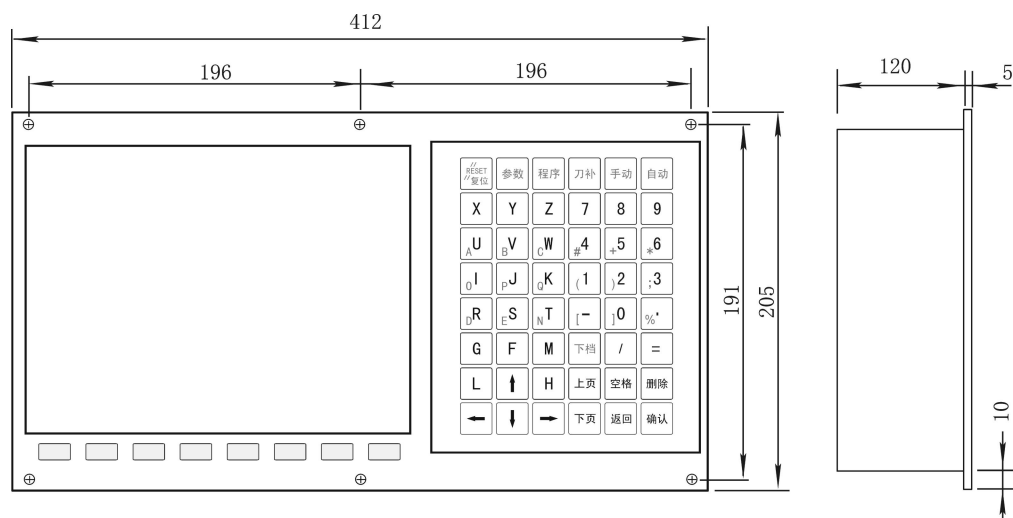
To ensure there is no strong magnet and current interference, keep away from inflammable, explosive and other danger materials.

Pay attention:

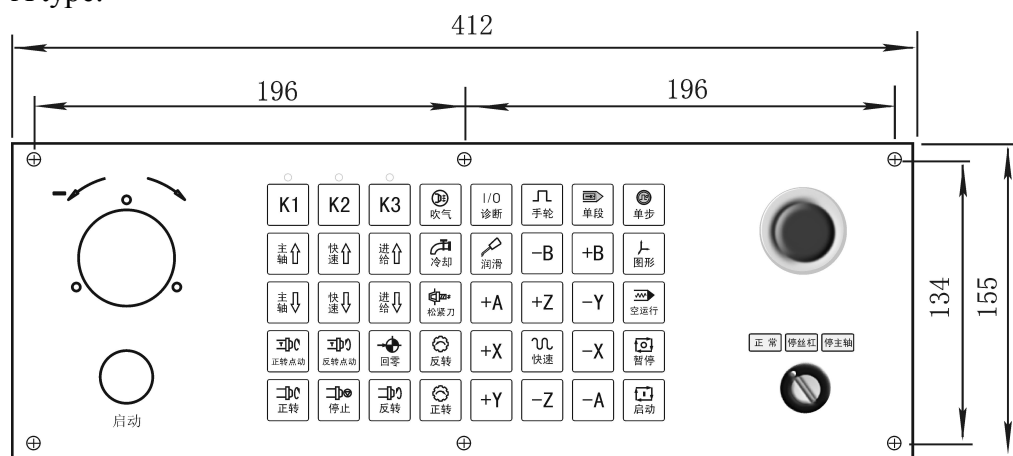
- 1. Must install in an electricity cabinet which is good for protect from lightning.**
- 2. Must install firmly to in order to avoid vibrating and losing.**
- 3. Don't install on the inflammable things or nearby to keep away from fire.**

2.5 System installation dimension

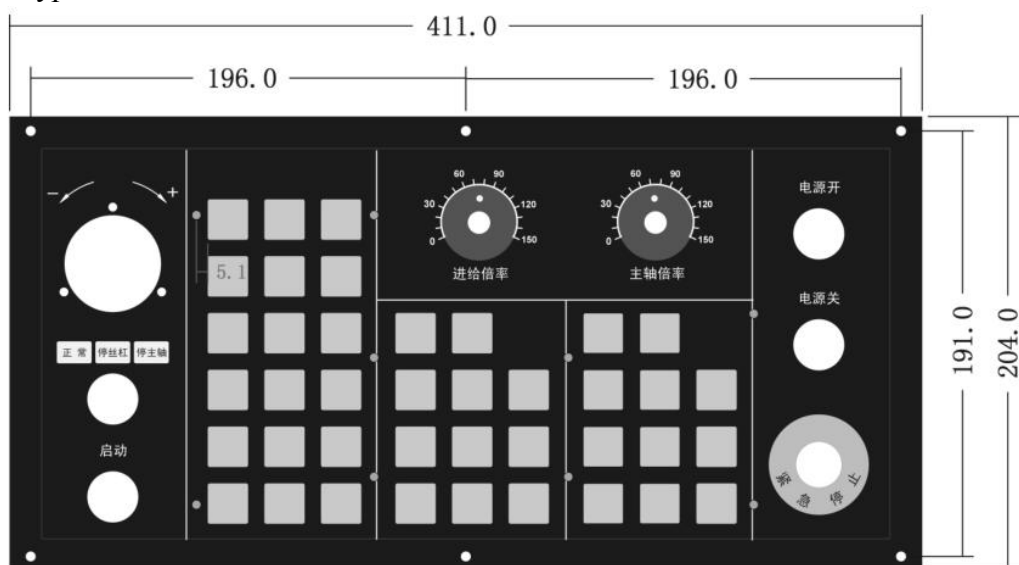
(A/B type: 325x250x120; C/E type: 400x245x120)



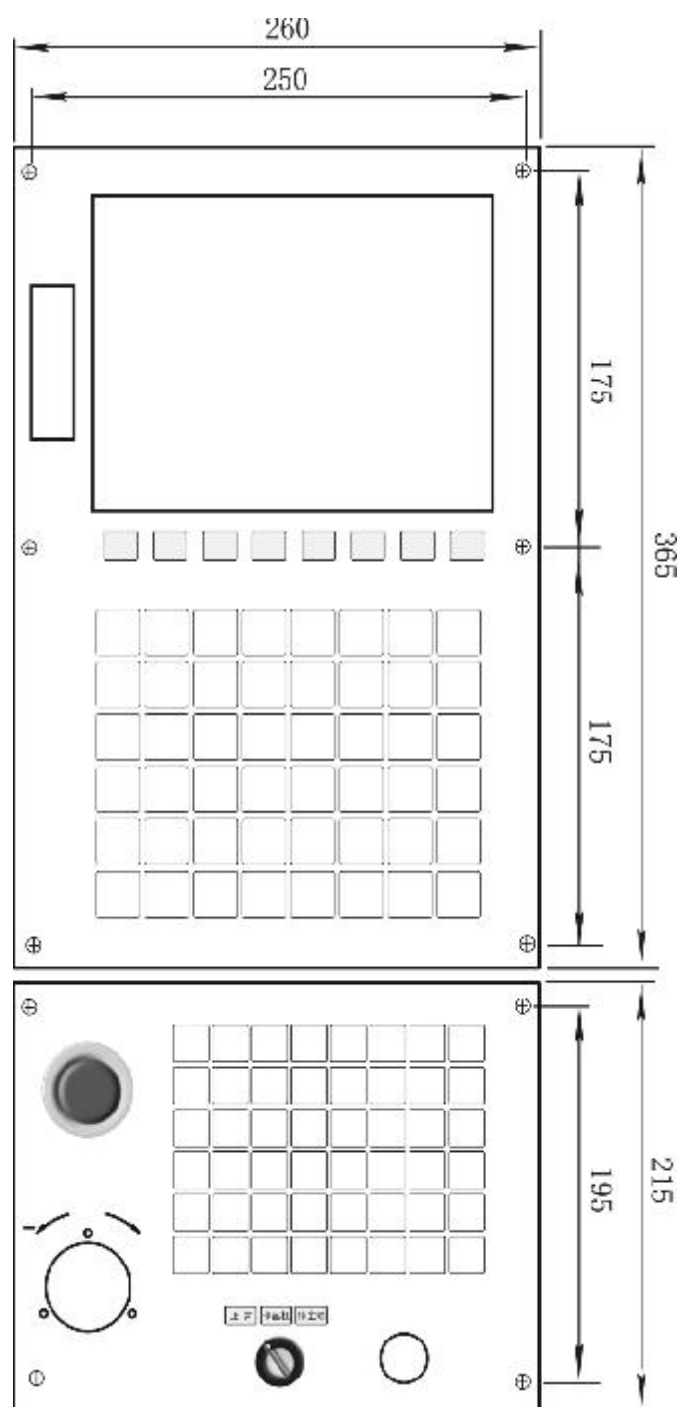
A type:



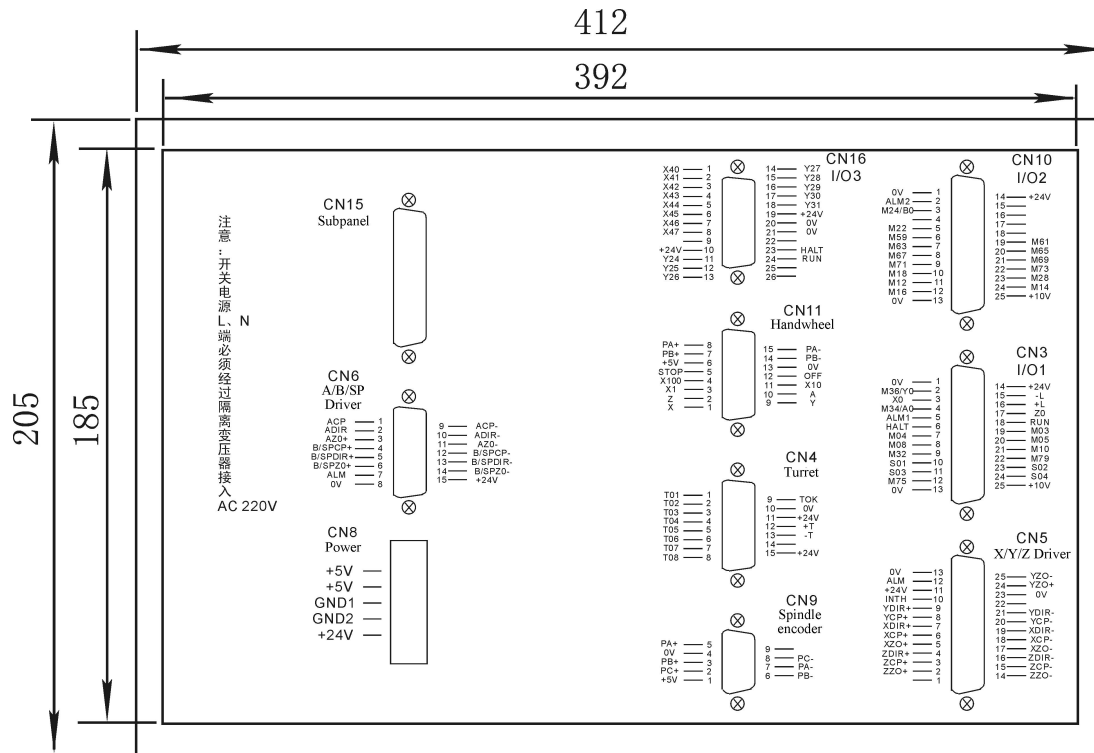
C type:



NEW1500TDC Controller



2.6 System rear view



T3	3	IN	T3 position signal	0V
T4	4	IN	T4 position signal	0V
T5	5	IN	T5 position signal	0V
T6	6	IN	T6 position signal	0V
T7	7	IN	T7 position signal	0V
T8	8	IN	T8 position signal	0V
TOK	9	IN	Turret lock signal	0V

System can control 1-99 tools, the electrical tool is 8 at most, default is 4. Press “C” to set the total tools in “redeem”, turret configuration is set in tool parameter.

P1: Activate the function of electrical tool to select electrical tool or tool post. [1 means Yes, 0 means No]

P2 parameter: 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.

P4: The maximum time for tool positive rotate. Failed to find tool position beyond the time, system will stop rotating tool and alarm. (Unit: second)

P5: Delay time after tool positive rotate to check the tool position signal. (ms)

P6: Delay time after tool stop. (ms)

P7: Tighten time of tool reverse rotate (ms).

P9: To set whether if it checks the key-locked switch after electrical tool reverse rotate to the position (already signal TOK), default is not to check.

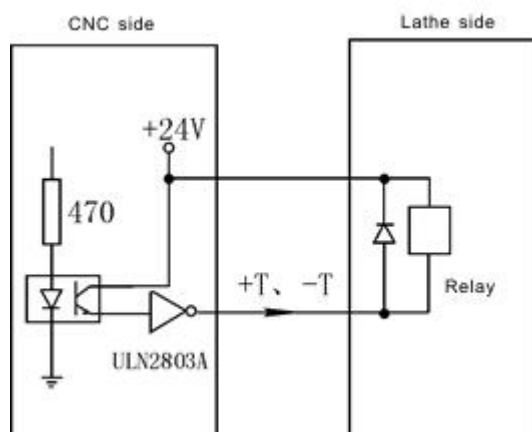
P20: Electrical turret type[1: electrical turret, 0: encoder turret], when set P20=0, the tool changing will be controlled by Macro program”ProgramTool”, it can be modified in Diagnosis interface, Press “F1 Control” to enter into program modification.

Pay attention:

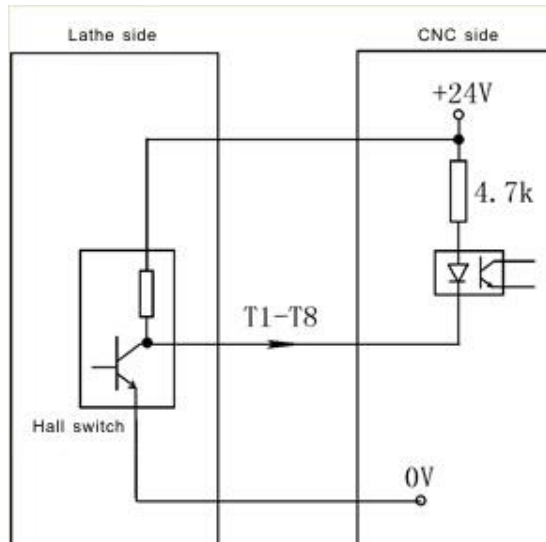
1. All the input or output is based on system, input is from exterior signal into system, the output is from system signal to external source.

2. When choosing the electrical appliance plate, +T and -T control single contact middle relay, user should install two AC contactors of +T and -T.

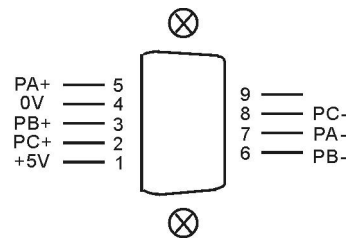
System output signal +T -T:



Tool input T1~T8 TOK:



2.7.2 Spindle Encoder CN9 Male socket

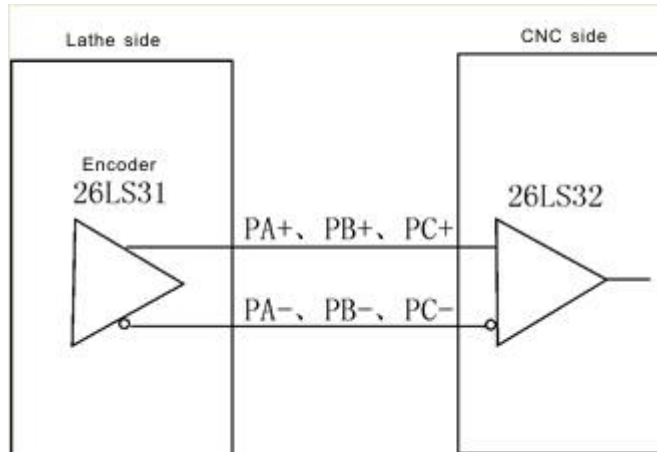


CN9 DB9(pin) spindle encoder				
signal	Pin	I/O	Function	Effective level
0V	4	OUT	0V	0V
+5V	1	OUT	+5V	+5V
PA+	5	IN	+A signal	5V
PA-	7	IN	-A signal	
PB+	3	IN	+B signal	5V
PB-	6	IN	-B signal	
PC+	2	IN	+Z signal	5V
PC-	8	IN	-Z signal	

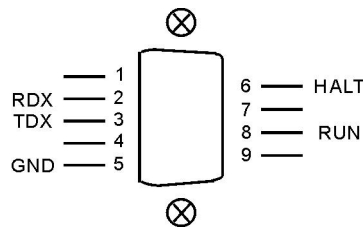
Pay attention:

1. The output signal of encoder is long-line output, the power supply is 5V.
2. The signal cable must use shielded twisted pair cable, the length is 20m at most.

The input signal of encoder PA PB PC:



2.7.3 Communication CN6 female socket

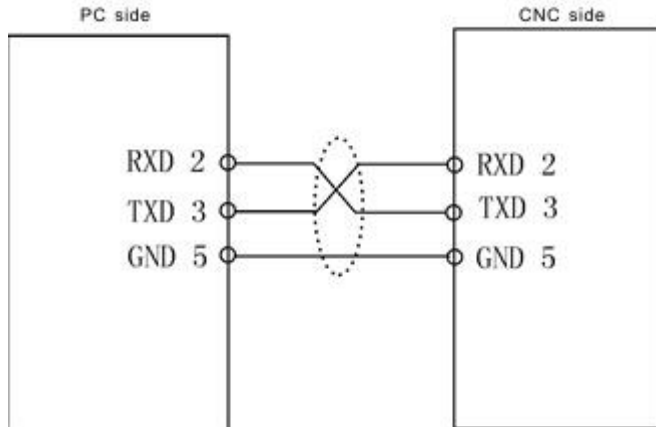


CN6 DB9(hole) RS232 communication				
signal	pin	I/O	function	availability
0V	5	OUT	0V	0V
RXD	2	IN	RXD	
TXD	3	OUT	TXD	
+5V	1	OUT	Power	+5V
LEFT	4	IN	Interfere switch left side	0V
HALT	6	IN	Pause	0V
RIGHT	7	IN	Interfere switch right side	0V
RUN	8	IN	Start	0V
STOP	9	IN	Emergency stop	0V

Pay attention:

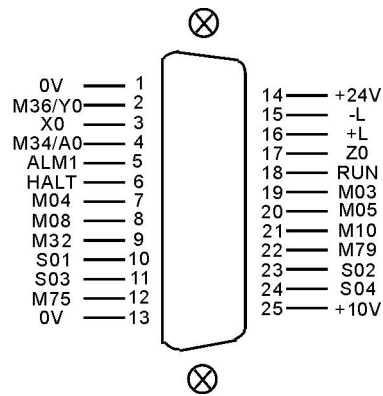
1. Must use the exclusive communication software to connect exterior PC for data communication.
2. The signal cable must adopt shielded twisted pair cable, the length is 10m at most.

The signal of CN6 connect to PC:



Pay attention: When PC programming, the files should be text files.

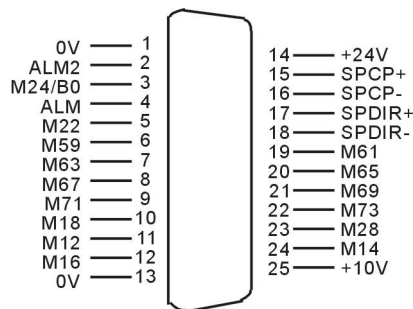
2.7.4 I/O 1 signal connection CN3 female socket(DB25)



CN3		I/O1 Machine signal DB25 female socket		
Signal	Pin	I/O	Function	Effective level
0V	1	OUT	Power ground	0V
+24V	14	OUT	24V power	+24V
M36/Y0	2	IN	Y axis home signal	0V
X0	3	IN	X axis home signal	0V
Z0	17	IN	Z axis home signal	0V
-L	15	IN	Negative limit	0V
+L	16	IN	Positive limit	0V
M34/A0	4	IN	A axis home signal	0V
ALM1	5	IN	Spindle alarm	0V
HALT	6	IN	External Pause button	0V
RUN	18	IN	External Run button	0V
M03	19	OUT	Spindle CW	0V
M04	7	OUT	Spindle CCW	0V
M05	20	OUT	Spindle stop	0V
M08	8	OUT	Cooling	0V
M10	21	OUT	Chuck tighten/loose	0V

M32	9	OUT	Lubrication	0V
M79	22	OUT	User defined Output 7	0V
S01	10	OUT	Spindle 1st gear	0V
S02	23	OUT	Spindle 2nd gear	0V
S03	11	OUT	Spindle 3rd gear	0V
S04	24	OUT	Spindle 4th gear	0V
M75	12	OUT	User defined Output 8	0V
+10V	25	OUT	The 1st analog output	0~10V
0V	13	OUT	Analog output ground	0V

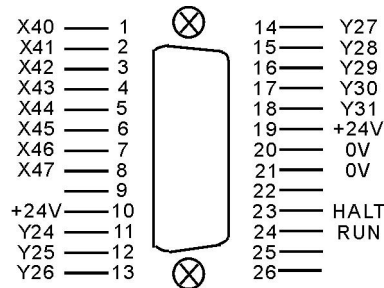
2.7.5 I/O 2 signal connection CN10 female socket(DB25)



CN10		I/O2 Machine signal DB25 female socket		
Signal	Pin	I/O	Function	Effective level
0V	1	OUT	Power ground	0V
+24V	14	OUT	24V power	+24V
SPCP+	15	OUT	Spindle pulse signal+	0V
SPCP-	16	OUT	Spindle pulse signal-	
SPDIR+	17	OUT	Spindle direction signal+	0V
SPDIR-	18	OUT	Spindle direction signal-	
ALM2	2	IN	Lubricant fault alarm	0V
M24/B0	3	IN	User defined Input 8 or B axis home signal	0V
M22	5	IN	Spindle orientation ready	0V
M59	6	OUT	Huff	0V
M61	19	OUT	User defined Output 1	0V
M63	7	OUT	User defined Output 2	0V
M65	20	OUT	User defined Output 3	0V
M67	8	OUT	User defined Output 4	0V
M69	21	OUT	User defined Output 5	0V
M71	9	OUT	User defined Output 6	0V
M73	22	OUT	User defined Output 7	0V
M18	10	IN	User defined Input 1	0V
M28	23	IN	User defined Input 2	0V
M12	11	IN	User defined Input 3	0V
M14	24	IN	User defined Input 4	0V

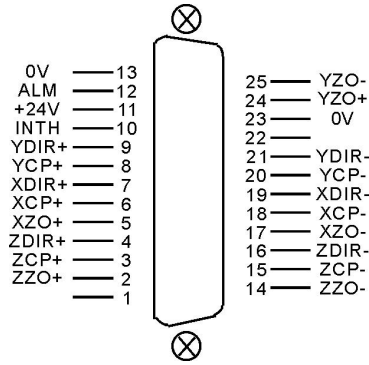
M16	12	IN	User defined Input 5	0V
+10V	25	OUT	The 2nd analog output	0~10V
0V	13	OUT	Analog output ground	0V

2.7.6 I/O 3 signal connection CN16 female socket(DB25)



CN16		I/O3 Machine signal DB26 female socket		
Signal	Pin	I/O	Function	Effective level
0V	20、21	OUT	Power ground	0V
+24V	10、19	OUT	24V power	+24V
X40	1	IN	Spare input 0	0V
X41	2	IN	Spare input 1	0V
X42	3	IN	Spare input 2	0V
X43	4	IN	Spare input 3	0V
X44	5	IN	Spare input 4	0V
X45	6	IN	Spare input 5	0V
X46	7	IN	Spare input 6	0V
X47	8	IN	Spare input 7	0V
Y24	11	OUT	Spare output 0	0V
Y25	12	OUT	Spare output 1	0V
Y26	13	OUT	Spare output 2	0V
Y27	14	OUT	Spare output 3	0V
Y28	15	OUT	Spare output 4	0V
Y29	16	OUT	Spare output 5	0V
Y30	17	OUT	Spare output 6	0V
Y31	18	OUT	Spare output 7	0V
RUN	24	IN	External Run signal	0V
HALT	23	IN	External Stop signal	0V

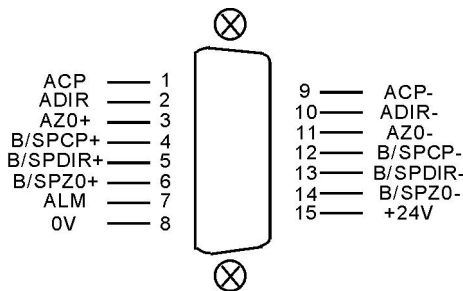
2.7.7 X/Y/Z Driver control signal connection CN5 male socket(DB25)



CN5 Servo driver control signal DB25 male socket				
Signal	Pin	I/O	Function	Effective level
XCP+/-	6/18	OUT	X axis pulse signal	5V
XDIR+/-	7/19	OUT	X axis direction signal	5V
YCP+/-	8/20	OUT	Y axis pulse signal	5V
YDIR+/-	9/21	OUT	Y axis direction signal	5V
XZO+/-	5/17	IN	X axis motor zero signal	5V
ZCP+/-	3/15	OUT	Z axis pulse signal	5V
ZDIR+/-	4/16	OUT	Z axis direction signal	5V
ZZO+/-	2/14	IN	Z axis motor zero signal	5V
YZO+/-	24/25	IN	Y axis motor zero signal	5V
ALM/0V	12/13	OUT	ALM Servo alarm	0V
+24V/0V	11/23	OUT	+24V/0V	24V
INTH	10	OUT	Servo alarm reset	0V

Attention:

When C axis is rotary axis, M800 is used to back encoder zero position, output M75 to switch spindle servo into position control mode, run M03/M04 will disable M75 signal, spindle servo switch into speed control mode. M881 command C axis spindle to orientate, output M61 signal, detect M22 as ready signal, then cancel M61 signal.

2.7.8 A/B Driver control signal connection CN6 female socket(DB15)

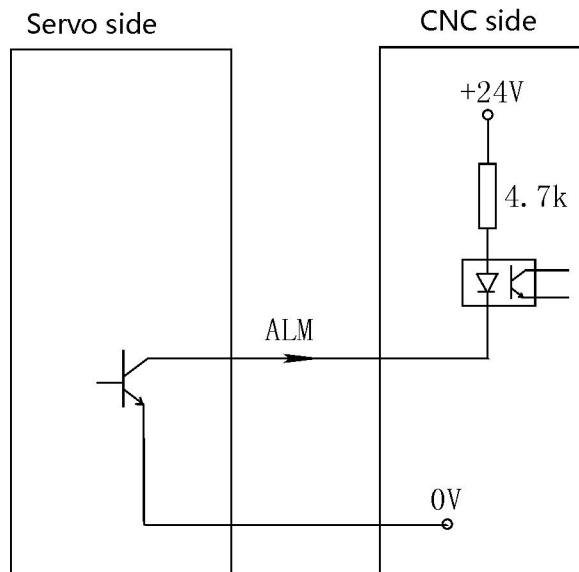
CN6 Servo driver signal DB15 male socket				
Signal	Pin	I/O	Function	Effective level
ACP+/-	1/9	OUT	A axis pulse signal	5V
ADIR+/-	2/10	OUT	A axis direction signal	5V
BCP+/-	4/12	OUT	B axis pulse signal	5V
BDIR+/-	5/13	OUT	B axis direction signal	5V
AZO+/-	3/11	IN	A axis motor zero signal	5V

BZO+/-	6/14	IN	B axis motor zero signal	5V
ALM/0V	7/8	IN	Servo alarm/0V	0V
+24V	15	OUT	+24V Power	24V

Attention:

1. driver signal cable must be twisted shielded pair, max length is 20m.
2. Servo alarm signal mode(NO/NC) is defined by Other parameter P17.

Servo Alarm signal diagram:

**2.7.9 External switch, electrical hand wheel CN11 male socket (DB15)**

CN11 DB15 Handwheel signal(male socket)				
Signal	Pin	I/O	Function	Effective level
0V	13	OUT	Signal ground	0V
+5V	6	OUT	5V power	+5V
PA+	8	IN	A signal positive	5V
PA-	15	IN	A signal negative	
PB+	7	IN	B signal positive	5V
PB-	14	IN	B signal negative	
STOP	5	IN	Emergency stop	0V
OFF	12	IN	Turn off	0V
H_X100	4	IN	x100 gear signal	0V
H_X10	11	IN	X10 gear signal	0V
H_X1	3	IN	X1 gear signal	0V

H_A	10	IN	A axis select signal	0V
H_Z	2	IN	Z axis select signal	0V
H_Y	9	IN	Y axis select signal	0V
H_X	1	IN	X axis select signal	0V

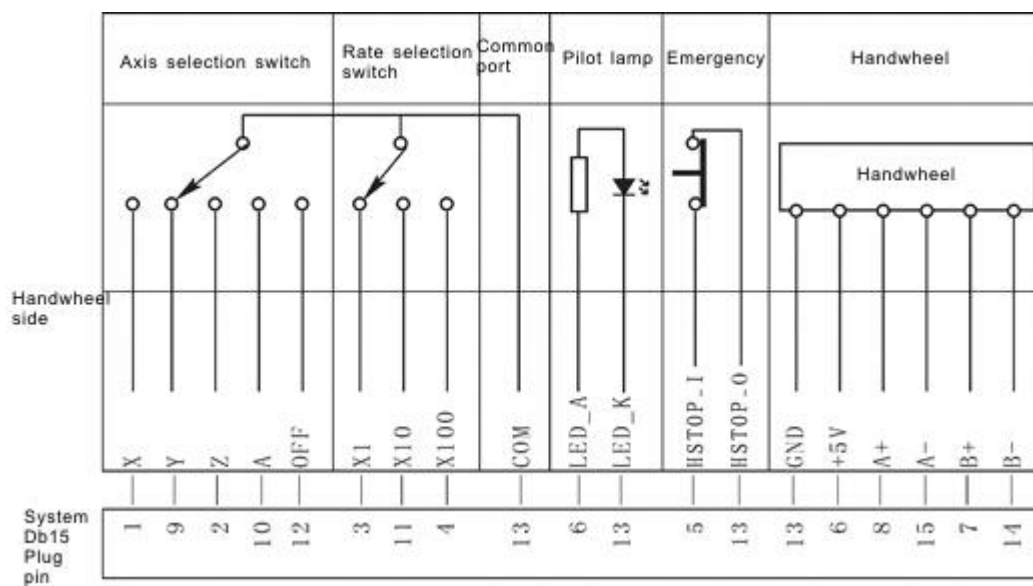
Pay attention: The inner power supply are all +5V of all signal, do not access voltage higher than 5V.

2.7.9.1 Usage for electrical handwheel(Manual pulse generator)

You can connect standard external handwheel when No.1 parameter in other parameter is 1 and can not use band switch to adjust spindle, feed and external stop running button,so No.1 No.2 parameter in axis parameter only set to be “0”

PA+ PB- PA+ PA- corresponding input signal of handwheel pulse A B.

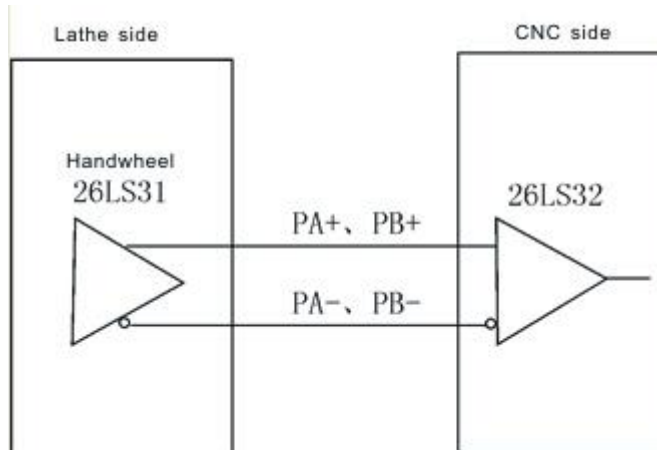
Handwheel contact diagrammatic as:



Pay attention:

1. The output signal of handwheel adopts line output, the power supply is +5V.
2. Just connect PA+ PB+ if adopt voltage output.
3. Manual pulse generator needn't "Enter" button, if there is a "Enter" button, use the line to short the ends of switch.
4. The inner power supply are all +5V of all signal, do not access voltage higher than 5V.

The input signal of handwheel:



2.7.9.2 Usage of terminal band switch behind sub panel

Signal	CN15 Pin	I/O	Function	Effective level
0V	29、39	OUT	Power ground	0V
+5V	11、26	OUT	5V power	+5V
PA+	41	IN	A Signal +	5V
PA-	12	IN	A Signal -	
PB+	27	IN	B Signal +	5V
PB-	42	IN	B Signal -	
STOP	44	IN	External EStop	0V
KRUN	14	IN	Run	0V
KLEFT	15	IN	Interfere switch left	0V
KRIGHT	30	IN	Interfere switch right	0V
DS0	8	IN	Spindle baud byte 1	0V
DS1	38	IN	Spindle band byte 2	0V
DS2	23	IN	Spindle band byte 3	0V
DS3	37	IN	Spindle band byte 4	0V
DK0	7	IN	Servo band byte 1	0V
DK1	22	IN	Servo band byte 2	0V
DK2	9	IN	Servo band byte 3	0V
DK3	24	IN	Servo band byte 4	0V
LRUN	43	OUT	Program run output	0V

the band switch works when Axis parameter P1=1, P2=1. DS0, DS1, DS2, DS3 are input signal which is for trimming gear of spindle speed, four position control, total 16 gears. VDK0 (OFF), VDK1 (X100), VDK2 (X10), VDK3 (X1) are input signal which is for trimming gear of feeding speed, four position control, total 16 gears.

2.7.9.3 Usage of external emergency button

STOP signal is the input signal of external emergency button, Other parameter P27 controls the emergency button is normal open or normal close.

2.7.10 Axis position feedback signal CN13 male socket(DB26)

CN13 Position feedback or linear scale signal DB26male socket				
Signal	Pin	I/O	Function	Effective level
+5V/0V	9/10	OUT	+5V/0V	
+24V/0V	26/23	OUT	+24V/0V	
DX+/DX-	25/24		RS485+/RS485-	
XA+/-	18/8	IN	X axis A signal	5V
XB+/-	17/7	IN	X axis B signal	5V
YA+/-	14/4	IN	Y axis A signal	5V
YB+/-	13/3	IN	Y axis B signal	5V
ZA+/-	16/6	IN	Z axis A signal	5V
ZB+/-	15/5	IN	Z axis B signal	5V
AA+/-	12/2	IN	A axis A signal	5V
AB+/-	11/1	IN	A axis B signal	5V
BA+/-	22/21	IN	B axis A signal	5V
BB+/-	20/19	IN	B axis B signal	5V

Other parameter P200-P220 are used to set this function. Press G in diagnosis interface to clear order position and feedback position, then reset over tolerance alarm.

Note:

1. encoder or linear scale output signal is long-cable output, power is +5V.
2. signal cable should be twisted shielded pair, no longer than 20m.
3. this is optional function

2.7.11 Absolute type controller setting steps

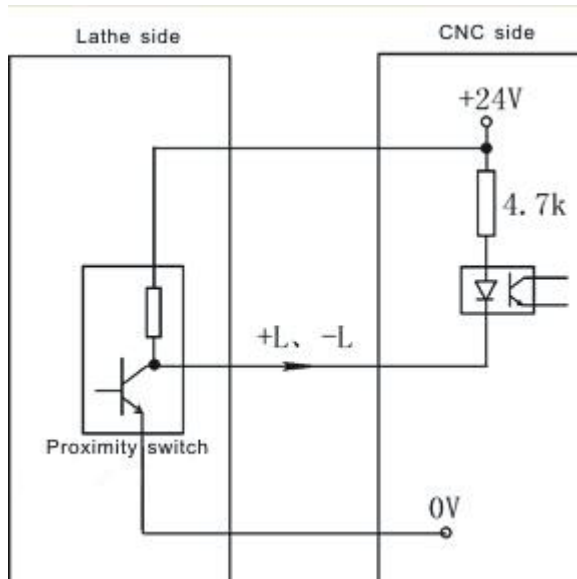
1. Turn on the power supply.
2. Set XYZA axis motor encoder type in Other parameter P300.
3. Set electrical gear in Axis parameter.
 Numerator = 10×reduction ratio
 Denominator = ball screw pitch
 Axis parameter P11=00000001; soft limit effectively;
 Axis parameter P23=11111011; home point with floating mode;
4. Set the other parameter in system
 P300=01111100; the absolute value function;
 P301=92; P302=91; P303=90;
 P304/P305/P306/P307=131072;
 P309/P310/P311/P312=(1/electronic gear)*10000000
5. check P56 parameter in driver(NEWKer driver)
 P1=1, enter the password;
 P56=1/2/3/4 corresponds to the X/Y/Z/A, such as the Z axis drive is set to 3;
 Press "Enter" for a while to save parameter in EP-status;

6. Run M500 in MDI, the small circle in front of machine coordinate should become green, that means read the absolute encoder data successfully;
7. Check coordinate feedback direction:
 - 1) move coordinate in manual mode and remember the machine coordinate value;
 - 2) Run M500(read all axis), or M501/M502/M503/M504(corresponds to the X/Y/Z/A axis) in MDI, check if the absolute value coordinate is still before and after running M501-M504, then run axis again and run M500 again, if coordinate are always change. Go to Other parameter P309-P312, change the sign of value in P309-P312.
Sample:
If X Servo motor is connect with ball screw directly. And ball screw pitch is 5mm.
 1. Axis parameter: electronic gear denominator P17=10, electronic gear numerator P18=5;
 2. Other parameter $P309=5/10 \times 10^7=5000000$;
A>Move X coordinate to 10.3456, then run M501 in MDI, check if coordinate is 10.34**, then move X coordinate to 25.1212, and run M501 in MDI again, to check if X coordinate is 25.12**. then move X coordinate to 30.4510, then run M501 in MDI again, and check if coordinate is 30.45**. then go to Step 8.
B>Move X axis to 10.3456, then run M501 in MDI, check if coordinate is 10.34**. if it is not. then move X coordinate to 25.1212, and run M501 in MDI again, to check if X coordinate is 25.12**. it changes less that 25 or bigger than 25. then change Other parameter=-5000000. then repeat Step A>;
8. Setting the zero point of machine: After finish step 4, move all axis to zero point, then set other parameter P314-P318, and input E into P314-P318, and press enter to clear the coordinates, finally, the current point is set as the zero point.
9. Manual move every axis to the limit position of machine to set the soft limit value in axis parameter;
10. Set the No.41 parameter in other parameter and back up the current system parameter

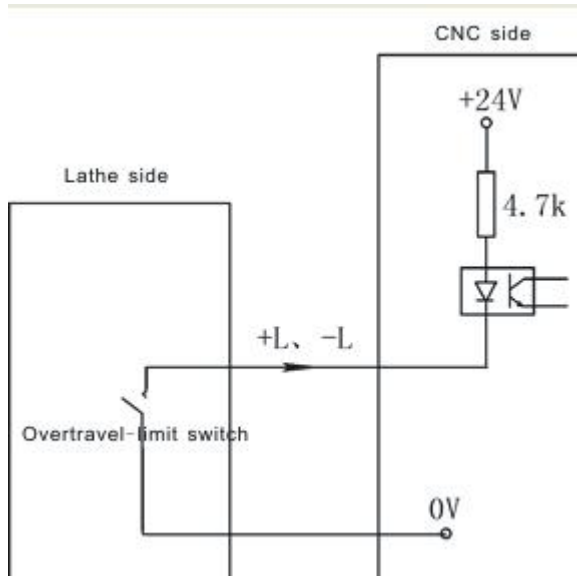
2.8 Machine electric element installation.

2.8.1 Limit position: take X Z axis is limited position positively as example

Model 1: NPN approach switch



Mode 2: general switch

**Axis parameter:**

21, XZ positive limited

[0 open, 1 close]

22, XZ negative limited

[0 open, 1 close]

Pay attention:

1. X Z axis limited shares a signal to always open or close together, positive limited and negative limited corresponding stand for +L and -L signal.

2. Could select our electrical appliance plate of lathe.

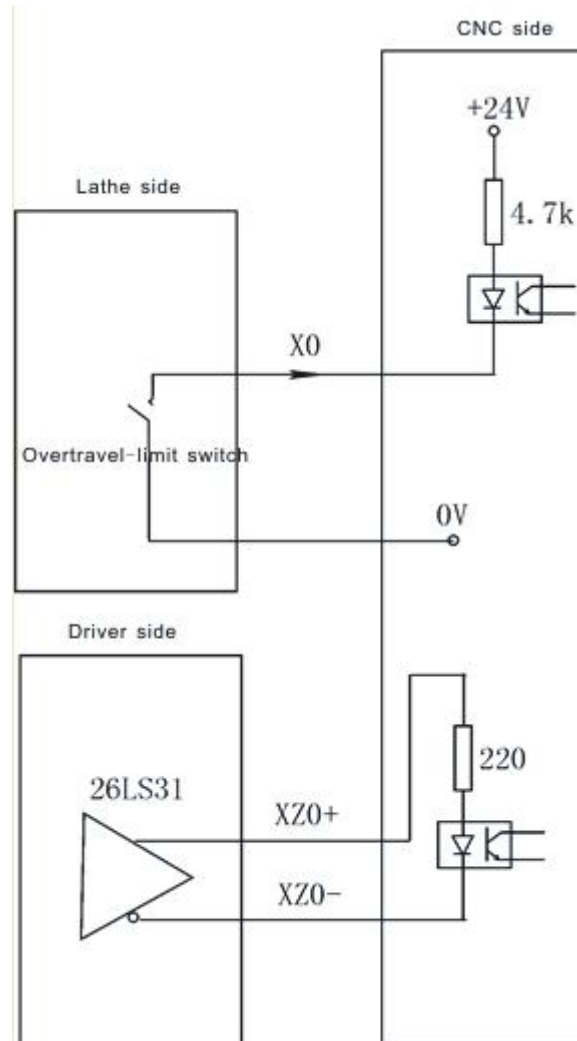
3. The system could define X0 Z0 to be limited input signal of X Z axis. X0 signal as the limited signal and home point of X axis, the same switch to control. Z0 signal as the limited signal and home point of Z axis, the same switch to control. No.37 No.38 function as follows:

No.21, X axis limited position [0 means normal open, 1 means normal close]

No.22, Z axis limited position [0 means normal open, 1 means normal close]

The function must copy our exclusive PLC software.

2.8.2 Lathe home point(reference point or 0 point): take X axis as example (the same as Z axis)



Backing to the home point

At the function of setting floating home point is invalid conditions, backing to the home point need to check approach switch signal and motor Z pulse signal. No.23 parameter in axis parameter is set to be "00000000".

No.26 parameter in axis parameter set the function of backing to the home points:

There are four ways for system to set backing to the home point when turn on the system:

1: means unnecessary, system doesn't prompt and no limits when turn on the system every time.

0: means prompt mode, system popup a dialog box to prompt user to process operation of backing to the home point, it has no limits.

8: means force mode, system popup a dialog box to prompt user to process operation of backing to the home point and not to process before running automatically, the system will import "Feeding axis doesn't back to the home point" and not to process program.

9: means super force mode, moving the feeding axis when turn on system every time, the system will popup a dialog box to process operation of backing to the home point, it will prompt "Feeding axis doesn't back to the home point" and not to process action if not to process operation of backing to the home point.

No.27 parameter in axis parameter set the checking signal mode of backing to the home point:

When it is 0: After hitting home switch when backing to the home point, run reverse to check the switch off and check 0 pulse signal of motor encoder.

When it is 1: After hitting home switch when backing to the home point, run reverse to check the switch off.

When it is 2: After hitting home switch when backing to the home point, continue running to check the switch off and check 0 pulse signal of motor encoder.

When it is residual value: After hitting home switch when backing to the home point, continue running to check the switch off.

The mode of backing home should accord to the circuit situation of equipment, at common conditions, it suggests to set to be 0 or 2, because if not to check 0 pulse signal of motor encoder, the accuracy can't be promise.

No.28 parameter in axis parameter set the direction and sequence of backing to the home point:

Every axis is to set separately. The parameter is positional parameter, D2 controls the processing direction of X axis, D4 is for direction of Z axis, 1 means negative, 0 means positive; D8 controls the sequence of X and Z, 1 means Z first, 0 means X first.

No.29 parameter in axis parameter set the type of home point switch:

Every axis is to set separately. The parameter is positional parameter, D0 position controls X axis, D2 position controls Z axis, 1 means always close, 0 means always open.

No.30, No.31 parameter in axis parameter is set to check the processing length of motor Z pulse when backing to the home point:

To set the scope of checking the motor encoder zero pulse signal after switch off when X(No.30) Z(No.31) axis backing to the home point. Unit: 0.1mm.

Pay attention: The parameter value must less than the distance of motor turns a round, otherwise could cause the wrong home point situation.

No.31, No.33 parameter in speed parameter set the speed of reaching to zero point switch when backing to home point (zero point):

The processing speed of reaching to home point switch when X(No.31) Z(No.33) axis backing to positive home point. Unit: mm/min. Numerical range: 20-500.

Pay attention: The parameter value influence with the accuracy of backing to the home point, the smaller the value the higher the accuracy. This value has been set which not to change anymore, otherwise it will affect the reference point too.

No.32 No.33 parameter in axis parameter set the offset of finishing back to the home point:

Used to set how much distance to offset before returning to the reference point when the axis backing to zero and checking the zero pulse signal of servo motor. Unit:

0.01mm. Numerical range:-99999~+99999.

The parameter value is related to the install position of lathe's home point and the lathe's coordinate.

Pay attention: After backing to the home point, the offset speed is determined by G00.

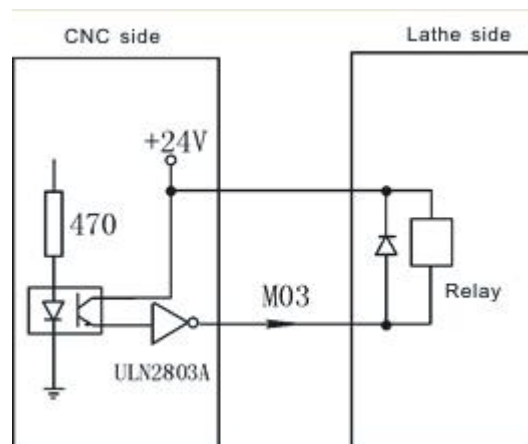
1. The speed reduce switch is also can use NPN switch.

2. Must consider the length of speed reduce when installing the speed reduce switch.

2.8.3 Switch signal control:

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

M03 signal control:



As the picture say, it will form a return circuit with 24V when system outputting M03, The intermediate relay is working and a group of normally open contact form a circuit with spindle rotation AC contactor.

All the low level 0V of output signal is effective.

Pay attention:

1. When the relays and others load, must connected with the diode to absorb the reverse current so as not to damage the system, if use the electromagnetic contactor, then plus resistive and capacitive spark circuit.

2. Chip ULN2803A corresponds output ports:

1). U28 corresponds M59 M61 M63 M65 M67 M69 M71 M73

2). U29 corresponds M03 M04 M05 M08 M10 M79 M32 M75

3). U30 corresponds +T -T S01 S02 S03 S04 LRUN INT

3. When user-defined signal M71/M70 M73/M72 is used for input signal of spindle chuck and thumbstall, it can't be the other user-defined. No.20 No.21 parameter in other parameter to set.

4. When user-defined signal M65 M67 M69 is used for input signal of "Emergency", it can't be the other user-defined. No.28 No.29 parameter in other parameter to set.

2.8.3.1 Spindle control (M03/M04/M05)

System output two spindles (First spindle, second spindle) controlling signal,

relative parameter as follows:

Axis parameter

No.7 parameter: Set the braking time of spindle, also the it's the output of the hold time, the time less the brake fast.

No.8 parameter: Set the braking signal is long signal 1 or short signal 0.

No.9 parameter: To set system whether checking spindle feedback signal of spindle position, the feedback signal is spindle encoder signal. To set the parameter value 1 means check; 0 means not to check.

No.10 parameter: To set feedback pulse numbers of spindle encoder turn a round, the value: Line number of encoder x 4.

No.50 parameter: Whether turn on the spindle or not when spindle shifting [1 means on, 0 means off]

No.51 parameter: The turning speed of motor when spindle shifting (1/100rpm)

No.52 parameter: The turning direction when spindle shifting (0 means positive, 1 means negative)

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

No.54 parameter: Turning time of low speed when spindle shifting (10ms)

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

Speed parameter:

No.8 parameter: To set the speed of spindle setting value at manual condition. Unit: rpm.

No.36 parameter: To set the highest speed of spindle, that's the turning speed of corresponding 10V instruction voltage.

No.37 parameter: To set the highest speed of spindle low gear(second gear) or the highest speed of second spindle, that's the turning speed of corresponding 10V instruction voltage. Unit: r/min

No.38 parameter: To set the highest speed of spindle (Third gear), that's the turning speed of corresponding 10V instruction voltage. Unit: r/min

No.39 parameter: To set the highest speed of spindle (Fourth gear), that's the turning speed of corresponding 10V instruction voltage. Unit: r/min

No.40 parameter: To set the highest speed of second spindle, that's the turning speed of corresponding 10V instruction voltage. Unit: r/min

Other parameter:

No.13 parameter: To set whether spindle and chuck is interlocking or not: 0 means they are separately; 1 means the spindle only start turning when chuck on. The thumb stall can't be use when the spindle is turning.

Setting parameter is related with the configuration of lathe and user's service condition, but consider for safe, suggest setting 1.

2.8.3.2 System lubrication control (M32/M33)

No.4 parameter in other parameter controls the function of lubricate automatically. No.6 parameter set the spacing time of lubrication (Unit: S); No.5 parameter in other parameter set the lubrication time (Unit: S).

2.8.3.3 Chuck and thumb stall control (M10/M11 M79/M78)

Chuck and thumb stall of this system control is related with parameter as follows:

Other parameter:

No.2 parameter: To set the controlling type of chuck, setting the parameter to set controlling logic when the lathe has automatic controlling chuck. Could realize to control inner and outer chuck (Inner: Chuck to center when M10; Outer: Chuck opening outward when M10). 1 means outer, 0 means inner.

No.15 parameter: To check chuck loosen or tighten enough when automatic control. 1 means check, check M12 when M10, check M14 when M11; 0 means not to check.

No.16 parameter: To set checking whether thumb stall goes forward or back enough when automatic control. 1 means check, check M18 when M79, check M28 when M78; 0 means not to check.

No.20 parameter: To set system controls chuck only need one signal (one-way valve) or two signals (two-way valve), this parameter is related with equipment of lathe.

M10 is just a output signal to control tautness of chuck when set to 0, system carry out chuck to tighten when M10 is effective, loosen chuck when M10 is invalid;

M10 and M71 corresponding control loose and tight of chuck when set to 1, system carry out chuck to tighten when relay M10 is effective, M71 is invalid, loosen chuck when M10 is invalid and M71 is effective. Output M10 when M10, output M71 when M11.

No.21 parameter: To set system controls thumb stall only need one signal (one-way valve) or two signal (two-way valve), this parameter is related with equipment of lathe.

M79 is just a output signal to control go forward or back of thumb stall when set to 0, go forward when M79 is effective, go back when M79 is invalid.

M79 and M73 corresponding control go forward and back of thumb stall when set to 1, the system carry out going back when M73 is effective, M79 is invalid. Output M79 when M79, output M73 when M78.

No.22 parameter: To set external button to control loose and tight of chuck(or foot switch), the signal is reciprocating, it means loosen once and then tighten once, reciprocating mode. No external button when set to 0; There is an external button to control chuck when set to 1, the signal is M16.

No.23 parameter: To set external button to control thumb stall goes forward and back(or foot switch), the signal is reciprocating, it means going forward once and then going back once, reciprocating mode. No external button when set to 0; There is an external button to control thumbstall when set to 1, the signal is M14.

No.24 parameter: To set the retention time when the output signal M10 M71 of chuck is short signal, set to 0 means the signal is long signal. Unit: S.

No.25 parameter: To set the retention time when the output signal M79 M73 of chuck is short signal, set to 0 means the signal is long signal. Unit: S.

Pay attention: M12 M14 M16 M18 and M28 are multiple functions signal, only choose one function to use.

2.8.4 System alarm signal: ALM, ALM1, ALM2, Door alarm M12 and Emergency

Other parameter:

No.7 parameter: To set the system whether to check the switching signal of protective door, no door switch when set to 0, there is a switch to control protective door when set to 1; Suggest to set 1 for safe.

No.8 parameter: To set the type of door switch, 0 means always open, 1 means always close.

No.17 parameter: To set the type of system checking the servo alarm signal (twelfth pin of CN5 ALM), 0 means always open, 1 means always close.

No.18 parameter: To set the type of system checking the spindle alarm signal of lathe (fifth pin of CN3 ALM1), 0 means always open, 1 means always close.

No.19 parameter: To set the type of system checking the alarm signal of lathe (second pin of CN19 ALM2), 0 means always open, 1 means always close.

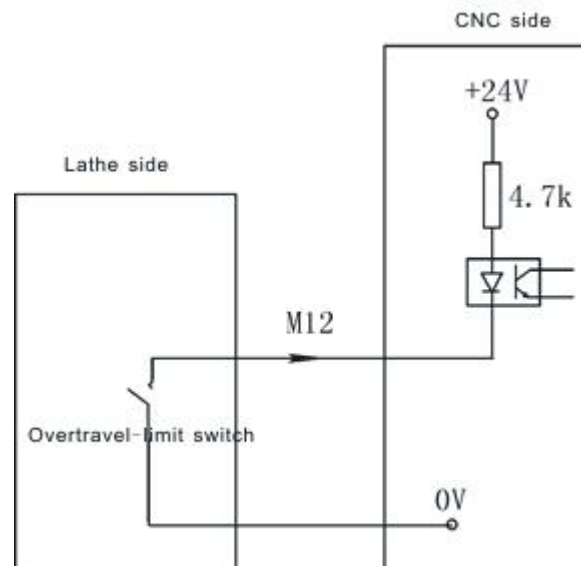
No.27 parameter: To set the Emergency always open or close of system CN11, suggest setting always close for safe.

Emergency STOP: Press “Emergency” when appearing emergent accident, the lathe will stop all actions and the screen of system shows “Emergency”. Wait for pressing up the button. Output M67 signal is effective (output alarm) when No.29 parameter in other parameter is effective. This output signal can be used to protect the lathe (Cut off power supply).

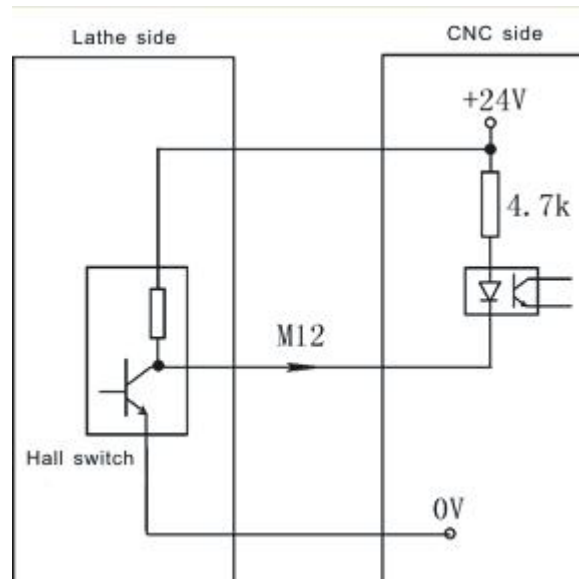
No.29 parameter in other parameter is effective when appearing alarm, the output signal M67 is effective.

2.8.5 User-defined output signal M12 (M14 M16 M18 M28 M22 M24, external “Run”, external suspend “HALT”, external “STOP” as the same)

M12 switch use overtravel-limit switch to wiring:



M12 switch also can use NPN as checking switch:

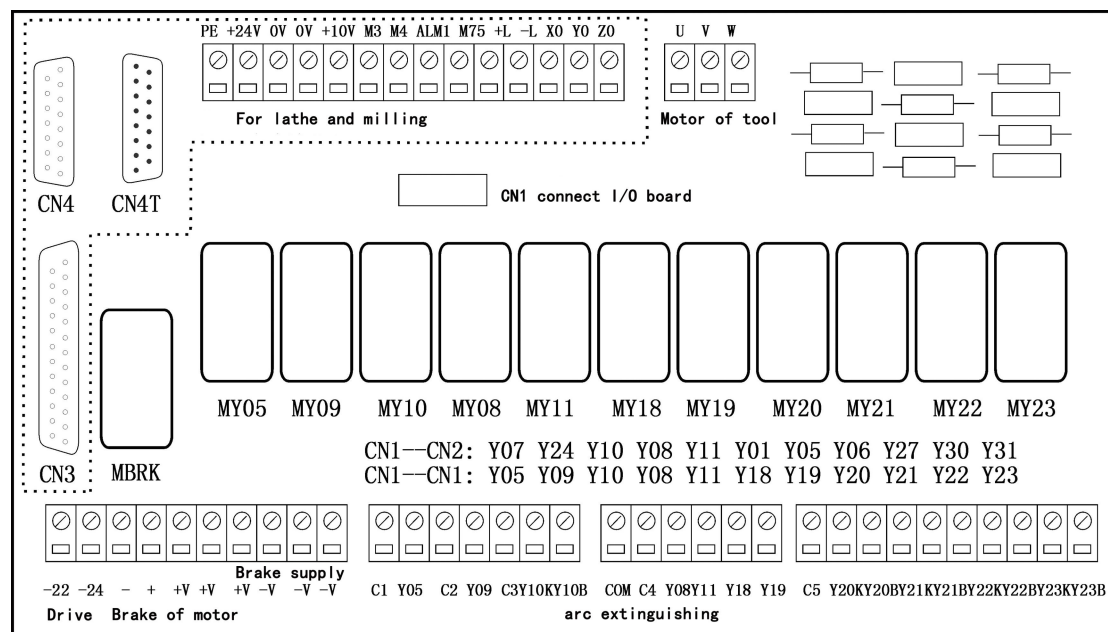


Pay attention:

1. M12 M14 M16 M18 and M28 are all multiple functions signal, only could choose one function to use.
2. All the low level 0V of output signal is effective.

2.9 Relay board of controller(optional)

Our company produces the electrical appliance plate of lathe to choose as follows.



There is a relay board produce by our company for your option:

1. CN1 connects to I/O board, 1 to 11 pins control relays;
Y05,Y09,Y10,Y08,Y11,Y18,Y19,Y20,Y21,Y22,Y23 correspond:
M71,M79,M10,M32,M08,+T, -T, S04,S03,S02,S01.
2. _22 connects to CN3-22 pin of driver, _24 connects to CN3-24 pin of driver;

3. -, + connect to motor with brake;
4. -V, +V connect with brake supply;
5. PE terminal must be connected to grounding bar of machine in case of interference;
6. Dotted box is applied to servo type controller: CN3, CN4 connect with CN3, CN4 of controller, CN4T connects to electrical turret signal socket, R21-R24 connect to pull-up resistor of electrical turret signal T1-T4.

2.9.1 Tool turret control

+T, -T must be controlled by external AC contactor.

C3 is the common port of +T, -T and M08.

2.9.2 Spindle control

C1 is the common port of M03 and M04.

C2 is the common port of M05, M5B is always closed.

2.9.3 Spindle gear control

C4 is the common port of S1 and S2, S1B and S2B are always closed.

C5 is the common port of S3 and S4, S3B and S4B are always closed.

Chapter 3 Operation

3.1 Summary

When using the NC system, just master the parameter of system, edit program, manual operation, auto operation. Then you can operate easily. There are some details to instruct hereinafter.

3.2 Operation panel and switches(Following shown: two types)

This controller includes main panel and sub panel, main panel is used to set parameter and edit program, sub panel is used to set tool and process operation.

3.3 Keyboard instruction

3.3.1 Rate increase

(1) Rapid ratio(G)

There are six gears in rapid ratio form 5% to 100%, by adjusting the key of rapid ratio is for the following instruction: G00,G26,G28,G611,G613, rapid feed fixed cycle, rapid manual feed.


(2) Feed ratio(F)


There are sixteen gears in feed ratio from 0% to 150%, by adjusting the key of feed ratio is for the following instruction:G01,G02,G03, the feed ratio of the fixed cycle and manual run effectively.

(3) Spindle ratio(S)





There are sixteen gears in spindle ratio from 5% to 150%, by adjusting the key of spindle ratio is for the speed of the first spindle.



















3.3.2 Switches

Switch	Functions
	Emergency stop Driver and motor stop immediately, turns off the spindle, coolant, waits for the rise of button, and initializes values

	<p>Interfere switch: When program runs or in manual state, it can make a real time adjustment of feed speed; Spindle percentage switch: In the process of spindle running, adjusts the speed accordingly.</p>
---	---

3.3.3 Other keys

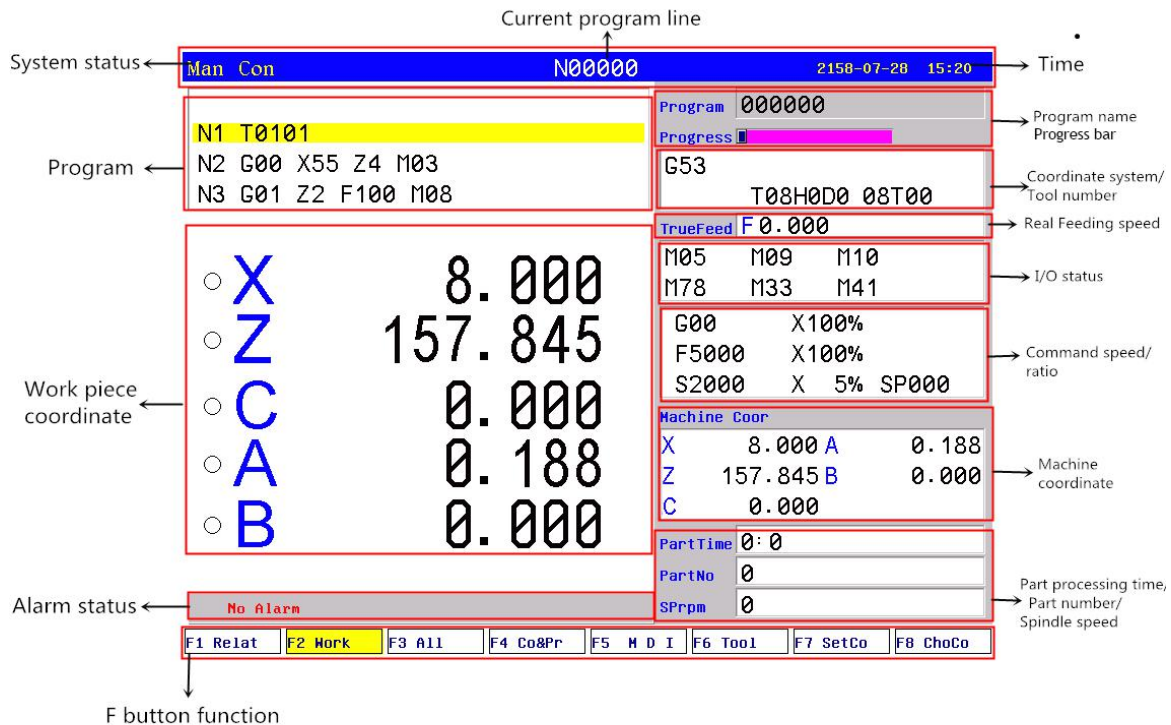
Keyboards	Functions
Letter key Number key	ABCSEFGHIJKLMNOPQRSTUVWXYZ123456789 . - : for program instructions, edition of parameter; number keys are used for inputting data and selecting sub-menu.
Edit key	“↑、↓、→、←、Del、PgUp、PgDn”for programming, direction keys can be used for selecting menu.
Function key	<p> “Esc” returning to upper level or stop a operation “Enter” selecting sub-menu and changing a newline “Del” delete program “program”entering program edition “parameter” entering parameter setting “diagnosis” entering diagnosis I/o function “manual” entering manual status “handwheel” for starting or stopping handwheel function “Tool” for confirming current tool ‘s position in machine tool coordinates system. “Redeem” for amending tool change errors “Auto” entering automatic status “MDI” entering MDI function “”selecting auto-coordinates/diagram machining “”for single segment or constant work “” for coordinates mode or diagram mode speed simulating “”for manual increment or manual continuous </p>

Control key	<p>“   ” spindle cw, ccw rotation</p> <p>“” coolant on/off</p> <p>“” for the shift between electric tool carrier and gang tool carrier</p> <p>“” for the shift between hand-driven continuous high speed and low speed.</p> <p>“” all axes return to home point</p> <p>“” for spindle chuck on/off</p> <p>“” for thumb stall on/off</p> <p>“” hand wheel gear selection</p> <p>“” adjusting spindle speed</p> <p>“” adjusting feed speed</p> <p>“” adjusting G00 speed</p>
Feed key	<p>    +Y +A-Y -A</p> <p>For X、Y、Z 、A axes direction feed</p>

3.4 Manual operation

The system adjusts one-level menu operation, intuitive, convenient, shortcut, prompt comprehensive information.

Powering the system is to enter the interface(Following shown)



The left part of the first line on the screen is the running mode of system(manual continuous, manual incremental, auto continuous, auto single block, auto idle running, hand wheel), middle part is the current line of processing program, right part is date and time.

The left top area is program part, the second line is current program line;

The left button area is work piece coordinate display;

The second area from bottom is the information display, like alarm;

The bottom area is the F1-F8 function keys.

The right area the display of program name, instructions, machine status and machine coordinate etc;

Menu operation


Press “program” to enter into program management, then edit, modify, check, delete, copy program;

Press “parameter” to enter into parameter management, then check and modify parameter of controller;

Manual mode is used to modify machine coordinate and management of machine auxiliary performance, tool setting, specify start point of work piece and manual process. Manual mode cannot used to run program.


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 increment”


(3)“S”: Set the speed of the first spindle. The range is from 0 to 99999, the max depends on the No.36 parameter in speed parameter.

(4)“I”: Modify the increment in manual increment

(5)“”: Press once to increase or decrease 10% feed speed when the No.1 axis parameter is 0, the range is from 0% to 150%,16 gears totally.

(6)“”: Press once to increase or decrease G00 or manual rapid override 20%.The range is from 5% to 100%,16 gears totally.

(7)“”: Press once to increase or decrease the spindle override 10% when the No.2 parameter in axis parameter is 0. The range is form 5% to 150%,16 gears totally.

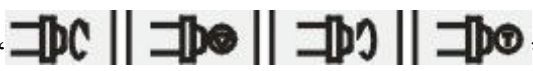
(8)“”: To switch cycle “0.001” “0.01” “0.1” or “0.1” “0.01” “0.001” in the hand wheel function.

(9)“Diagnosis”: Enter the diagnosis of input or output.

(10)“Setup”: To set a value(G54-G59) in workpiece coordinate(G54-G59);Use “MDI” to set G54-G59 in lathe coordinate(G53).

(11)“Auto”: Select automatic mode.

(12)“Manual”: Select manual mode.


(13)Spindle controlled: “”Controlling spindle on clockwise, counterclockwise, stop, correspond to instructions M03,M04,M05. When No.56 parameter in the axis parameter is “8”then press “spindle on counterclockwise” means counterclockwise inching turning.


(14)“Cooling”: Coolant on or off correspond to instructions M08,M09.

(15)“Chuck”: Chuck tightens or loose correspond to instructions 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) “Tail stock”: Tail stock tighten or loose correspond to instructions M79, M78.


(18) “Switch manual continuous or increment”: Press  to manual continuous or increment, it displays I=XXXX.XXX when it is manual increment.

(19) “Back to home point”: Press  and X or Z, the X or Z axis goes back to the home point automatically; Press “0”X axis firstly and then Z axis; Press “Esc” to cancel the construction. The speed controlled by No.31 No.33 parameter in speed parameter, the direction is determined by No.28 parameter in axis parameter.

(20) “Tool carrier controlled”: Press  to change next tool automatically if it is gang tool carrier; After changing next tool it will be stop if it is electric tool carrier;

Which tool has changed is going to be redeem. Press “T” and number to change tool directly

(21) “Coordinates feed”: Press “↑ ↓ ← →” correspond to feed A axis and Z axis’s positive or negative direction.

(22) “Switch speed”: Press  to switch the speed to system speed which is changed by No.1 No.2 parameter in speed parameter when it is in coordinate feed, loosen it that will be the previous speed. If set the speed higher than the speed in parameter, it will be the set speed to feed.

(23) “Switch coordinates’ display”: Press “PgUp” or “PgDn” to switch the display which correspond to “relative” “absolute” “machine”.

(24) “Switch coordinates’ display”: Press “PgUp” or “PgDn” to switch the display which correspond to “relative” “absolute” “machine”.

(25) “Partno clear”: Press Del and Enter to clear quantity of processed work piece.

(26) User-defined “K1”: Turn on/off Y24

(27) User-defined “K2”: Turn on/off Y25

(28) User-defined “K3”: Turn on/off Y26


(29) “Incremental coordinate”: Press “Setup” to fix or set 0 after select “relative” coordinate.

(30) search center point: convert into G54-G59, press”K”. according to prompt information, controller will calculate value and save the offset of G43-G59.

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

3.4.2 Manual continuous


Continuous operation is to press the time as the basis, Press to feed, up tp 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 the speed to No.1 No.2 parameter value in speed parameter. If set the speed higher than the speed in parameter, the feed speed will be No.1 No.2 parameter in speed parameter times rapid override.

In order to facilitate the user single axis cutting in the manual function, setting the manual speed in manual status. Press “F” and input the speed.



When the hard limit point beyond positive and negative feed running axis two direction at, stop the feed and prompt to feed reverse direction.(the same as hereinafter)

The manual maximum speed is limited by No.6 parameter in speed parameter, when setting the speed is higher than the value of parameter, then will be the No.3 parameter.

When No.38 other parameter is 8, “” is change into a switch, press once to turn on (no more to always press), press again to turn 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 Back to lathe home point (reference point)

There are two ways to back to home point in this system, not only the switch for home point, but also can set floating point, the methods as follows:

Home switch installed:

Back to home point operation is to feed every axis to lathe's home point position in turn. When the parameter of feeding axis which back to home point is 0, the axis of coordinate detects the home point and return to the pulsing signal of “Zero”, the data of lathe's coordinate will be 0 automatically.

Switch on the power supply of the system, release alarm and the button of emergency after the CNC is power off, the need to back to home point to set lathe's coordinate correctly.

Instruction:

1. The system requires for backing to the home point every time when it is power on, the requirement can be set by No.45 parameter in axis parameter, it can be prompt or force;
2. The way and type of detecting signal can be set by No.46 parameter in axis parameter, so detect the switch of home point is effective, also detect the Z pulsing signal of electrical motor after detecting the switch of home point (precision higher), detect forward or reverse for Z pulsing signal of electrical motor.
3. The direction for backing to home point can be set by No.47 parameter in axis parameter, D2 D4 correspond to X Z axis, 0 is forward, 1 is reverse.
4. The sequence of X Z back to home point can be set by No.47 parameter in axis parameter, X is first when D8 is 0, Z is first when D8 is 1.
5. The type of the switch for home point can be set by No.48 parameter in axis parameter, D0 D2 correspond to X Z axis, 0 is always on, 1 is always off.
6. The maximum length of detecting Z pulse of electrical motor can be set by No.49 -No.53 parameter in axis parameter, the value must less than the pulse of electrical motor run a cycle.
7. The shifting distance after backing to home point can be set by No.54 - No.58 parameter in axis parameter, rapid move coordinate to the value of parameter after backing to home point.

Without home switch:

To set floating point to make sure, turn on corresponding function of floating point by No.39 parameter in axis parameter, setting No.40 -No.44 parameter to make sure X axis' and Z axis' floating point, the home point of lathe.

The steps to set floating point as follows:


1. Setting the No.39 parameter in axis parameter to set the axis which is starting up floating point. For example: Turn X axis on is "00001000". (Z axis is "00100000" turn both of them on is 00101000.)
2. Moving X axis to designated position so that set floating point.
3. Press "Parameter", "Axis parameter" and select No.30 parameter, "Enter", popup a dialog box of X axis' floating point coordinate. Import the value of setting lathe coordinate.

If it is 0, the lathe coordinate of X axis now is the home point of X axis. The lathe backs to this position every time when backing to the home point.

If it is 15, the current lathe coordinate of X axis is 15.000, the distance to lathe's home point is 15mm.

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

Operation for backing to the home point:

At the manual condition, press " "and select X Z axis to back to the home point in dialog box. Set the No.47 parameter in axis parameter to "1" to make Z axis back to the home point first. At the absolute and relative coordinate condition, the cycle will turn to green in front when backing to the home point successfully, defeat otherwise.

If stop in the process, press "Stop" or "Reset" to stop backing to the home point.


Pay attention:

Every time to power up the system must back to the home point to make sure the accuracy of lathe process. The system power off unusually or in an accident, it must back to the home point, otherwise could cause trouble.

3.4.5 MPG(Hand wheel)

Two types: hand held and panel, No.1 parameter in other parameter to set.

Panel

Press "handwheel" and "Z" or "X" to select an axis, " " to adjust the gear.

Hand held

Press "handwheel" and operate the switch of axis selection to select an axis, operate the axis and switch of handwheel override to adjust the gear.

Instruction

The handwheel is mainly used for "Tool", the speed and the handwheel feed of one measure is related to rotate the handwheel fast or low. The speed is not too fast best when the system cooperate with stepping motor.

Handwheel's pulse generator speed to be lower than 200r/min(The handwheel to 100 pulse a cycle), the handwheel's acceleration is controlled by No.26 parameter in speed parameter(the bigger the faster). The maximum speed is controlled by No.29(X axis) No.31(Z axis).

Handwheel is of no effect in auto-coordinates diagram machining, it only works in working coordinates.

3.4.6 Tool set

Because no tool is the same when using multiple tools machining so need to identify them previous the value of the redemption, that is to carry out redeeming. Tooling is actually move the tool to the workpiece surface at this point, the point's actual measurement of values import directly into the system the system calculates the deviation and save to the corresponding tool offset register automatically.

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

Plan A(suggest)

- (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) Z axis exits(X axis can't move), stop spindle.
 - (5) Measure the diameter of workpiece(cylinder or bore).
 - (6) Press "Tool", then "X", "Enter" and import the above value of measurement into dialog box, press "Enter" to confirm.
 - (7) Use the same method to cut end surface of workpiece.
 - (8) Measure the end surface of workpiece and spindle chuck (Z=0) into the distance.
 - (9) Press "Tool", then "Z", "Enter" and import the above value of measurement into dialog box, press "Enter" to confirm.
- The second tool is already done(T02).Repeat (1) ——(9) to make others tools get done.

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 import 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 workpiece and spindle chuck (Z=0) into the distance.
- (12) Press "Z" and import the above value of measurement into dialog box, press "Enter" to confirm.

The second tool is already done(T02).Repeat (1) ——(12) to make others tools get done.

The difference between two methods:

Plan A(suggest)

- (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 import value's and lathe coordinate value's difference will be saved to the corresponding cutter compensation by system automatically. So the inaccuracy of setting tools can be modified correctly by corresponding cutter.

System to each tool independently of each other, each has its own coordinate system, so each tool can "Tool" anytime and the tool is destroyed in the process is only the tool.

Instruction

- 1. When a group of tool to be used for two or more parts processing requires the working coordinate to achieve the overall shifting tool group. So, make sure the value of workpiece coordinate before setting tools. Methods of operation are as follows:**

- (1) Select a tool.
- (2) Press "MDI" to select corresponding coordinates (54-59).
- (3) Using manual continuous to cut a bit of cylinder or bore on workpiece.
- (4) Z axis exits(X axis can't move), stop spindle.
- (5) Measure the diameter of workpiece(cylinder or bore).
- (6) Press "Setup", "X" and "Enter", import the value of measure, press "Enter".
- (7) Use the same method to cut end surface of workpiece.
- (8) Measure the end surface of workpiece and spindle chuck (Z=0) into the distance.
- (9) Press "Setup", "Z" and import the value of measure, press "Enter".

The import value's and Tool coordinate value's difference will be saved to the corresponding parameter by system automatically, corresponding workpiece coordinate is been set now. It's done after finishing setting tools as the above. It's done that the code in first line to execute selection of the corresponding coordinate.

- 2. Just setting one of the tools after the lathe crashing or loss of step, every tool is ok.**

The method: In the G53 status, switch "Tool" into "Setup" as the above operations.

3.5 Auto operation

Auto refers to processing the editing program of workpiece. This system can start at arbitrary point, and also can start at arbitrary line or with arbitrary tool. Starting arbitrary line or with arbitrary tool must use absolute coordinate to edit the program. Auto operation can't move the manual coordinate.

Running program selection: In the program interface, press “↑ ↓” to move the cursor to a program which is going to be carry out, press “C” to select the program to carry out automatically.

Switch the display of coordinate: Press “PgUp” or “PgDn” to switch the display which correspond to “relative” “absolute” “machine”.

3.5.1 Automatic process

“Single or continuous”: Press “” to switch cycle.


“Continuous”: The program continue to execute every program segment(program line) to end or the instruction of stop to stop.

“Single ”: The program just execute one program line and end, wait another operation or press “Run” again to execute one next program line.

“Coordinate or figure”: Press “” to switch cycle.

“Automatically coordinate”: The axis of coordinate will display with value.

“Automatically figure”: The axis of coordinate will display with a figure. There are two kinds of figure, horizontal lathe and slant-bed lathe, No.3 parameter in tool parameter to control.

“”: The program is speedy simulate, the axis of coordinate can't move.

3.5.2 Start from any program line or tool number line

A. start from real line

At the automatic process condition, press “—” to pop up a dialog box, input line number, press “Enter” to confirm, the line will be the line to run.

Pay attention:1. The line is the actual line in the program, not the “N” stand for the line. The system process to the line you import with a speed which is set by No.8 parameter in speed parameter(G01/G02/G03), then process the program normally.

2. The line of default is the line of suspend the program last time, to facilitate user's operations.

3. At the interface of coordinate to use “N” to search line and press “Reset” to back to the beginning of program.

B. Start from marked line

The system has a function to run at the marking line. At automatic process condition, press “N” to pop up a dialog box to import the marking line, press “Enter” to confirm. Press “Run” to process program at the line you import(mark).

Pay attention:

The line is not the actual line, is the “N” stand for the line. The system process to the line you import with a speed which is set by No.8 parameter in speed parameter(G01/G02/G03), then process the program normally.

C. Start from any tool number line

The system has a function to run at some tool. At automatic process condition, press “G” and the number of tool to run(only the number of tool, not the number of

compensation; Like: 0304, just import “03”), press “Enter” to confirm. Press “Run” to process program at the tool you import.

Pay attention: The system process to the line you import with a speed which is set by No.8 parameter in speed parameter(G01/G02/G03), then process the program normally.


3.5.3 Start program

Press “Auto” to switch to automatic mode to process program, two methods as follows.

- (1) Press “Run”
- (2) Switch on the Run of external signal.

3.5.4 Stop processing program

Five methods as follows to stop:

- (1) The instruction of program M00 M01 M02 M30 M20.
- (2) Press “” to run a current line and stop.
- (3) Intervention switch in the middle or right.
- (4) Switch on the Halt of external signal.
- (5) Press “Reset” to stop all the actions of program.(Like spindle, tools and others)

3.5.5 Real-time control in automatic mode

(1) Intervention switch: For suspend feed coordinate and stop spindle. Left, middle, right 3 gears totally.


Left: Feeding coordinate and spindle is not limited.


Middle: Stop feeding, spindle is not limited.


Right: Stop feeding and spindle.

Stop here means suspend, turn left the switch to continue process; Press “Reset” to exit automatic process and stop processing, the program line is going to back to the first of the processing program.


At the manual mode condition, spindle is not limited by intervention switch, only by the button.

- (2) “” : Press once to increase or decrease 10% feed speed when the No.1 axis parameter is 0, the range is from 0% to 150%,16 gears totally; When the No.1 axis parameter is 1, external band switch takes in control, Adjust the speed of process arbitrarily in the process according to the different situation.

(3) “” : Press once to increase or decrease G00 or manual rapid override 20%.The range is from 5% to 100%,16 gears totally. Adjust the rapid override arbitrarily according to the different situation.

(4) “” : Press once to increase or decrease the spindle override 10% when the No.2 parameter in axis parameter is 0. The range is form 5% to 150%,16 gears

totally. When the No.1 axis parameter is 1, external band switch takes in control, Adjust the speed of spindle arbitrarily in the process according to the different situation.

(5) Stop in the process: At the continuous mode in process condition, press  to stop running after executing a current program line, wait for operating.

(6) Suspend in the process: Turn the intervention switch right or middle and switch on external stop signal of Halt, the processing program will stop; Press “Reset” to exit automatic process mode and the program line is going to back to the first of the processing program.

(7) Keep feeding: When the process is suspending, press “Manual” to keep feeding automatically, also can adjust the coordinate, press “Auto” and “Run” to run to the point of suspend automatically to end.

(8) Exit process: Press “Reset” when processing, suspending or keep feeding.

(9) enable inner macro program during processing. For example: if X7 is valid, start ProgramUser9, add PLC as following:

```

X07 M122
- - | | - - | |------(S)M300      ;if X7valid then set M300
X07 M300 M122
- - | | - - | | - - | |------(S)M1014    ;if X7valid, reset current program;
M300 M122 X07
- - | | - - | / | - - | / |------(S)M1069    ; enter into manual mode;
M300 M122 M01 X07
- - | | - - | / | - - | / |------(S)M1079    ; execute ProgramUser9;
                                |--(R)M300      ; reset M300;

```

M300: intermediary register, PLC cannot appear the same M300;

M1014: reset current program;

M1069: manual mode;

M1070-M1079: corresponding to ProgramUser0-9.

3.5.6 MDI Mode

At the manual or automatic coordinate conditions, press “M” to get into the processing mode of MDI. Processing a program line that you import in “MDI”, press “Esc” to give up and exit when importing, press “Run” to carry out the program line that you import.

3.5.7 MPG(hand wheel) mode

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

Pay attention: The acceleration, deceleration and maximum speed of running hand wheel are controlled by Speed parameter P25-P33, P49, User parameter P23, use the default acceleration, deceleration and the speed of G00 when the

parameter is set to be invalid.

3.5.8 DNC Function

The storage space of user is 32Mbit in this system, use DNC to process when the processing program is greater than 32M or the remainder storage space. Switch on RS232 or USB to realize the function of DNC in this system.

A. Instruction of RS232-DNC

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

Pay attention: 1. The baud rate is related to operational environment when using serial port to send files.

2. The communication cable can't more than 10 meters length.

3. Only the dedicated communication software of this system can send program in user's computer. To set the sending speed of PC as the NC, defeat otherwise.

B. Instruction of USB-DNC

USB-DNC is realized by U-disk, switch on U-disk and system, select program to execute in U-disk.

Press “B” to open U-disk in program interface, select corresponding program to press “C” to execute program, press “Auto” to get into automatical mode and press “Run” to process the program.

Pay attention: 1. Don't unplug U-disk in the process of USB-DNC, defeat processing otherwise.


2. Back to the system program interface from U-disk interface after finish USB-DNC.

3. After selecting the program, it is best to press “P” to compile once to make sure the program is right before executing program of USB-DNC.


3.6 Safe operation and Alarm

3.6.1 Emergency stop



Press “” when emergency accidents happening, the system will stop all the actions of lathe and shows “Emergency stop” on the interface. Wait for the button up. M67 imports effective signal when No.29 parameter in other parameter to be set effectively.



Press “” in the process or running lathe, system coordinate and lathe’s position may change, make sure the system coordinate again before processing, it is best to carry out operation of backing to the home point to make coordinate same as the lathe’s position.

The button can be external which is controlled by No.27 parameter in other parameter to set it always open or close.

3.6.2 Reset system

Press “Reset” to stop current operation in anytime when the system is running, especially stop all the actions of lathe(spindle, tools and so on) in automatical or manual mode, but the coordinate won’t lose, so needn’t to back to the home point.

3.6.3 Alarm

The screen shows error information and twinkles when the lathe has alarm, the program is stop running, the coordinate stop moving, check the reason for alarm and clear troubles to run again. The signal M67 is effective when No.29 parameter in other parameter is “1”.

- (1) **X and Z axis are limited positive** : X or Z axis reaches positive limit switch.
- (2) **X and Z axis are limited negative**: X or Z axis reaches negative limit switch.
- (3) **Spindle and inverter (frequency converter) alarm**: The alarm signal of spindle is effective. (ALM1)
- (4) **Oil alarm**: Oil shortage alarm is effective.(ALM2)
- (5) **X and Z axis’ driver alarm**: The alarm signal of servo drivers is effective. (ALM). Press “B” to import INTH signal to reset the servo drivers in diagnosis mode.
- (6) **No.5 alarm for door switch**: The alarm signal of M12(door switch) is effective.
- (7) **+5V is under voltage**: Supply voltage is low, +5V of the system is low.
- (8) **Emergency stop**: Press the button of emergency stop.

3.7 Parameter operation

At any status conditions, press “parameter” to enter the status to set the parameter. Parameter in this system includes “User parameter” “Speed parameter” “Axis parameter” “Tool parameter” “Other parameter” “Coordinate” “Password”, 7 kinds totally.

Man Con		N00000		2013-08-23 11:26	
1,Cycle G71/G72		50		Program 99.TXT	
2,Cycle G71/G72		10		Instruction code	
3,G71G72G173 in		0		G53	
4,G71/G72/G73 d		20		T0000	
5,G71/G72/G73 d		20		Machine Status	
6,G73 cutting t		1		M05 M09 M10	
7,G73 X rough t		10		M78 M33 M70	
8,G73 Z rough t		10		G00 X100%	
9,G74/G75 defau		50		F30 X100%	
10,G74/G75 defa		10		S50 X100%	
11,G76 finish t		1		Machine Coord	
12,G76 quit len		10		X 0.000 A 0.000	
13,G76 thread t		60		Z 0.000 B 0.000	
				C 0.000 SP360	
				PartTime 0: 0	
				PartNo 0	
				SPrm 0	
No Alarm					
User	Speed	Axis	Tool	Other	Coord
				PASSWD	CANCEL

Press “F1-F7” correspondingly to enter corresponding interface after enter the parameter interface, press “↑ ↓” to select the number of parameter and press “Enter” to prompt a dialog box to import data and press “Enter” again to fix parameter successfully.

Note: :

1. Value of all bit parameters are counted from right to left, different bit corresponds to different function.

Format: D9|D8|D7|D6|D5|D4|D3|D2|D1|D0

Instructions of parameter as follows:

3.7.1 User parameter (processing)

Pay attention: More details for instructions refer to chapter 4.

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

Used to set default X feeding value in G71 if without specification in program, (radius value), unit is 0.01mm.

Used to set default Z feeding value in G72 if without specification in program, unit is 0.01mm.

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

Used to set default X retract value in G71 if without specification in program, (radius value), unit is 0.01mm.

Used to set default Z retract value in G72 if without specification in program, unit is 0.01mm.

★、3,G71G72G73[D2:auto machining,D3:G74 X fast feed,D4:G74 Z fast feed,D5:auto direction,D6:auto point,D7:G70 no home,D8:tool radius]

D2=1 means G71/G72/G73 instruction do finishing process automatically;

D3=1 means in G74 instruction, X axis moves by G00 speed;

D4=1 means in G75 instruction, Z axis moves by G00 speed;

D5=1 means G71/G72/G73 instruction automatically analyse feed and retract direction, otherwise the sign of XZ axis finishing process allowance and G73 rough processing amount is effective, then controller will judge direction from program sign;

D6=1 means define feeding start point from finish process track, otherwise the first feed point of G71/G72 start from positioning point;

D7=1 means G70 do not back to start point;

D8=1 means when G71/G72/G73 has tool radius compensation, rough process allowance should be bigger than tool radius.

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

Used to set default X axis finish process allowance(diameter) in G71/G72/G73 when program does not define.

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

Used to set default Z axis finish process allowance(diameter) in G71/G72/G73 when program does not define.

★、 6,G73 cutting times

Set default G73 cutting times.

★、 7,G73 X rough thickness(10um)[X axis radius]

Used to set default X rough process amount in G73 when it is not define in program.

★、 8,G73 Z rough thickness(10um)

Used to set default Z rough process amount in G73 when it is not define in program.

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

Used to set default Z feeding value in G74 when it is not define in program.

Used to set default X feeding value(radius) in G75 when it is not define in program.

★、 10,G74/G75 default backward distance(10um)[X axis radius]

Used to set default Z retract value in G74 when it is not define in program.

Used to set default X retract value(radius) in G75 when it is not define in program.

★、 11,G76 finish turn times

Used to set G76 circle times when not defined in program.

★、 12,G76 quit length(1/10 lead)

Used to set G76 default chamfer retract length when not defined in program, retract length=screw lead*0.1.

★、 13,G76 thread tooth angle(degree)[0~180°]

Used to set G76 default thread angle when not defined in program.

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

Used to set minimum cut depth of G76. unit is 0.01mm

★、 15,G76 finish turn remaining(10um)

Used to set G76 default finish turning reserve value when not defined in program, unit is 0.01mm.

- ★、16,X program mode[1 mean Radius,0 mean Diameter]
Used to set X axis program mode, 0 means radius mode, 1 means diameter mode.
- ★、17,Running program need Sp run[1 mean Yes,0 mean No]
Used to set interlock between spindle and program.
- ★、17-1,Running program need Sp M10[1 mean Yes,0 mean No]
Used to set interlock between spindle chuck and program.
- ★、18,Set M20 the time of auto-running[Negative mean immensity loop]
Used to set M20 circle times, negative value means endless loop.
- ★、19,Set part count
Used to display or set M20 circle times.
- ★、20,G92 quit length(1/10 lead)
Used to set G92 default chamfer retract value when not deined in program, retract distance=screw lead*0.1.
- ★、21,G01/G02/G03 line delay(ms)[>100]
Used to set delay time between G01/G02/G03 lines.
- ★、22,G00 line delay(ms)[>100]
Used to set delay time after G00.
- ★、23,Handwheel acceleration[50-100]
The smooth acceleration constant when using handwheel, The range is 50-100, smaller value corresponds the faster acceleration, but more vibration.
- ★、24,G76 Q[8 mean thick forward number]
Used to set function of Q in G76, “8” means Q is rough turning feeding times, refere to G76 instruction.
- ★、33,M03/M04/S detect the spindle speed(0:M69 relay, 8:encoder feedback)
M03/M04/S code detect if spindle speed reach already. 0 means detect M69 relay, 8 means detect spindle encoder.
- ★、34,The error (RPM) of the spindle speed is detected by the encoder feedback.
The telorance error of spindle speed feedback from encoder.
- ★、200,system screen protect times[>=2minutes]
When under main interface and no dialog, and if P18>2 minutes, controller will enter in screen protection, any key to quit.
- ★、201,G92/G76 delay time(ms)[>100]
The delay time before zero pulse when process screw.
- ★、203,Whether strict inspection G41/G42 (34818:No,6326274:Yes)
The controller inspects overcut of G41/G42 strictly.
- ★、210,Enable graphics display area(8>manual,0:Automatic)
Enable graph displaying.
- ★、211,X axis negative end
X axis negative end of graph displaying;
- ★、212,X axis positive end
X axis positive end of graph displaying;
- ★、213,Y axis negative end
Y axis negative end of graph displaying;
- ★、214,Y axis positive end

- Y axis positive end of graph displaying;
- ★、 215,Z axis negative end
Z axis negative end of graph displaying;
 - ★、 216,Z axis positive end
Z axis positive end of graph displaying;
 - ★、 230,Select executive program through input
(+4...+128,+256...+32768:X26-X31,X16-X23)
Executing program by input points, for example, if P230=+4+8=12, when X26 or X27 is effective to choose the program HIDEFILEX26 or X26 or HIDEFILEX27 or X27.
D2=1, means +4, if X26 is effective, then execute Program X26 or HIDEFILEX26;
D3=1, means +8, if X27 is effective, then execute Program X27 or HIDEFILEX27;
D4=1, means +16, if X28 is effective, then execute Program X28 or HIDEFILEX28;
D5=1, means +32, if X29 is effective, then execute Program X29 or HIDEFILEX29;
D6=1, means +64, if X30 is effective, then execute Program X30 or HIDEFILEX30;
D7=1, means +128, if X31 is effective, then execute Program X31 or HIDEFILEX31;
D8=1, means +256, if X16 is effective, then execute Program X16 or HIDEFILEX16;
D9=1, means +512, if X17 is effective, then execute Program X17 or HIDEFILEX17;
D10=1, means +1024, if X18 is effective, then execute Program X18 or HIDEFILEX18;
D11=1, means +2048, if X19 is effective, then execute Program X19 or HIDEFILEX19;
D12=1, means +4096, if X20 is effective, then execute Program X20 or HIDEFILEX20;
D13=1, means +8192, if X21 is effective, then execute Program X21 or HIDEFILEX21;
D14=1, means +16384, if X22 is effective, then execute Program X22 or HIDEFILEX22;
D15=1, means +32768, if X23 is effective, then execute Program X23 or HIDEFILEX23;
 - ★、 231,"Delete" mode[0:backward deletion,1:Forward delete]
 - ★、 232,Check whether the spindle before tapping [18:Yes,0:No]
Check spindle home position before tapping;
 - ★、 233,G06 Circle teaching function[0:No,1:Yes]
 - ★、 234,Program back function[+8:Yes]
Use handwheel to run program forward or backward;
 - ★、 235,Automatically generate comments when teaching [0:Yes,1:No]
 - ★、 307,M18xx/M28xx/WAT alarm time(ms)[>=10]
The max waiting time for M18xx/M28xx/WAT code, if ove the time, controller will alarm.
 - ★、 400,Translate DXF file to G code[1:Seqencing,4:Start point seqencing ,8:No seqencing]
 - 1) if P400=1, controller will sequence end to end;
 - 2) if P400=4, controller will sequence end to end, and judging distance between ends to tool start point by parameter, and choose to start tool from nearby end;
 - 3) if P400=8, controller will not sequence. When generate DXF file from AutoCAD, please pay attention to draawing sequence, because DXF file save date according to drawing sequence, the g code generated by controller will also execute according to drawing sequence.
 - ★、 401,Translate DXF file to G code X-axis coordinate

Tool start point X coordinate of DXF file.

- ★、402,Translate DXF file to G code Y-axis coordinate

Tool start point Y coordinate of DXF file.

- ★、500,G74 equal to ProgramG No.[101-170(101-150Modeless,151-170Mode)]
- ★、501,G81 equal to ProgramG No.[101-170(101-150Modeless,151-170Mode)]
- ★、502,G82 equal to ProgramG No.[101-170(101-150Modeless,151-170Mode)]
- ★、503,G83 equal to ProgramG No.[101-170(101-150Modeless,151-170Mode)]
- ★、504,G84 equal to ProgramG No.[101-170(101-150Modeless,151-170Mode)]

3.7.2 Speed parameter

- ★、1,X-axis's G00 speed(mm/min)

- ★、2,Z-axis's G00 speed(mm/min)

The most speed of G00, the maximum: 30000.

- ★、3,Manual maximum feed speed(mm/min)

The max speed in manual mode in manual mode.

- ★、4,Auto Maximum feed speed(mm/min)

The max speed in manual mode in auto mode.

- ★、5,G01/G02/G03 default speed(mm/min)

When the speed is not defined in the first interpolation command(G01/G02/G03), program will run in the P8 default speed. Maximum: 5000.

- ★、6,Null run speed(mm/min)

- ★、7,Feed axis's manual speed(mm/min)

The speed when program does idle/null running. Maximum: 240000.

- ★、8,Spindle's manual speed(rpm)

Spindle speed in manual mode.

Note: Press S in main interface, also can modify the speed.

- ★、9,Take-off speed(mm/min)

Beginning speed when axis starts to move.

- ★、10,Maximum mutation speed(mm/min)

- ★、11,Limit G1G2G3 axis speed[1 mean Yes,0 mean No]

Set max feed axis speed is limit during interpolation.

- ★、12,X-G1G2G3 max speed(mm/min)

- ★、13,Z-G1G2G3 max speed(mm/min)

XZ max speed during G1G2G3 interpolation.

- ★、14,X acceleration

- ★、15,Z acceleration

XZ axis acceleration time constant, the bigger value sets, the faster acceleration is. Range:1-99999.

Note: the value depends on machine configuration, incorrect setting may lead to machine malfunction. Usually, the bigger load, the smaller value. If installed with stepper motor, it can not be over 15000.

- ★、16,Auto run acceleration[>=500]

Auto run acceleration constant, rang:1-500. when the valuse is between 1-500, auto run acceleration speed depends on this parameter. Otherwise, refers to axis acceleration speed parameter in user parameter. This parameter is used to extinguish

manual and auto acceleration speed, only when the acceleration speed in two mode has big difference, will this parameter work. Normally it is ineffective.

★、17,Handwheel acceleration[500--30000]

The acceleration speed when use handwheel, range is 500-32000, bigger value means faster acceleration.

★、18,Run program Handwheel acceleration[>500]

The acceleration speed when use handwheel in auto mode, range is 500-32000, bigger value means faster acceleration. When P27<500, it will be ineffective.

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

The idle running speed when use handwheel in auto mode. Ineffective when lower than 10.

★、19-1,Run program Handwheel G00 start speed(mm/min)[>5]

★、20,X-Handwheel limit speed(mm/min)

★、21,Z-Handwheel limit speed(mm/min)

Used to set max axis speed when use handwheel, range: 100-max manual speed. The value of P20-P21 depends on machine configuration. It should not be over 4000 in order to run safely. Works only when P29-P33a is bigger than 100.

★、22,Make thread Z acceleration

★、23,Make thread X acceleration

★、24,Servo motor screw thread X axis Back speed

★、25,stepper motor thread X retract start speed(mm/min)

★、25,stepper motor thread X retract max speed(mm/min)

★、27,Acceleration type[0 mean line,8 mean curve]

Used to set acceleration type of axis, 8 means acceleration is curve type(S shape), 0 means linear type.

Depending on driver types, usually stepper motor is curve type, servo motor is linear type.

★、28,Curve initial acceleration[>=10]

Initial acceleration constant in curve acceleration type, bigger value means faster acceleration

★、29,Curve acceleration with second-order[>=10]

The second acceleration constant in curve acceleration type, bigger value means faster acceleration.

★、30,Curve max acceleration[>=500]

max acceleration constant in curve acceleration type, bigger value means faster acceleration.

★、31,X go home rampit speed(mm/min)

★、32,X go home reverse speed(mm/min)

★、33,Z go home rampit speed(mm/min)

★、34,Z go home reverse speed(mm/min)

The speed of X axis going home sensor in backward direction, range: smaller than X G00 speed.

The speed of Y axis going home sensor in backward direction, range: smaller than X G00 speed.

★、35,The lower limit value of the spindle speed of G96 under constant line speed control(rpm)

★、36,Spindle first max speed(rpm)

The max speed of spindle, when run max speed, controller will output 10V analog signal.

★、37,Spindle second max speed(rpm)

★、38,Spindle third max speed(rpm)

★、39,Spindle forth max speed(rpm)

★、40,Second Spindle max speed(rpm)

The max speed of the second spindle, when run max speed, controller will output 10V analog signal.

★、41,reverse compensation mode(0 mean A; 8 mean B)

When D3=0, A type: When interpolation compensation of arc reversal, the speed of compensation is effective with reversal interpolation value, the bigger value, the faster speed to make sure tool will not pause, but the compensation speed can't over 10000mm/min;

+8(D3 bit), B mode: When interpolation compensation of arc reversal, the compensation speed is specified by No.49-No.51 parameter.

+1(D0 bit): Length compensation mode is B mode, otherwise is A mode. A mode is always compensating in Z axis, B mode is specified by G17/G18/G19, G17 compensate Z, G18 compensate Y, G19 compensate X.

+2(D1 bit): The fixed loop mode is B mode, otherwise is A mode. A mode is always compensate in Z axis, B mode is specified by G17/G18/G19, G17 compensate Z, G18 compensate Y, G19 compensate X.

+4(D2 bit): IJK is coordinate from the end point to centre of circle in arc programme, plus 4 means the IJK is to the end point in G02 G03, otherwise is to start point.

★、42,mode B reverse compensation speed(mm/min)

★、42-1,mode B reverse compensation Beginning feed speed(mm/min)[>10]

★、42-2,mode B reverse compensation acceleration(mm/min/s)[>10]

★、43,Handwheel stop speed(mm/min)[>100]

Minimum speed of spindle before rotate CCW during tapping;

★、49,speed Mode(1 Yes,0 No)

To set if do speed pre-processing to short line.

★、51,Follow the tapping knife when the spindle speed(rpm)[>1]

Minimum speed of spindle before rotate CCW during tapping;

★、52,When tapping spindle backlash compensation (pulse)

Spindle backlash compensation before rotating CCW.

★、53,Follow the the tapping cutter withdrawal before reversal(um)[10-5000]

★、54,Tapping back speed(mm/min)[>=60]

★、58,Forcedly limit drop speed critical(mm/min)

Used to set the speed when motor start to reduce speed in hard limit. Servo

needn't reduce, set as 1 normally.

- ★、 60,Smooth running function(+4:Manual;+8:Handwheel;+16:Program)
- ★、 61,Manual smooth running time normal[2-500]
- ★、 62,Handwheel smooth running time normal[2-500]
- ★、 63,G01/G02/G03 smooth running time normal[2-500]
- ★、 65,Manual enhancement smoothing processing time constant [2-50]
- ★、 66,Handwheel enhancement smoothing processing time constant [2-50]
- ★、 67,Program enhancement smoothing processing time constant [2-50]
- ★、 68,Enhanced smoothing process rating [1-9] (the bigger the smoother the more smooth)
- ★、 101,X-Beginning feed speed(mm/min)[>1]
- ★、 102,Y-Beginning feed speed(mm/min)[>1]
- ★、 103,Z-Beginning feed speed(mm/min)[>1]
- ★、 104,4-Beginning feed speed(mm/min)[>1]
- ★、 105,5-Beginning feed speed(mm/min)[>1]

Used to set initialize speed when feed axis doing acceleration, and also the end speed of deceleration speed. If feed axis running speed is lower than P101-P105, there is no acceleration or deceleration. If running speed is higher than P101-P105, acceleration starts from P101-P105 speed.

Attention: the value of P101-P105 is relative to machine configuration, wrong setting may lead to machine malfunction.

Usually, if installed with stepper driver, less than 100, if installed with servo driver, less than 500.

- ★、 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]
- ★、 114,4-Jump speed at continuous track(mm/min)[>1]
- ★、 115,5-Jump speed at continuous track(mm/min)[>1]

Used to set break incremental speed of each axis when axis speed changes during mutli axis interpolation in order to ensure resultant endpoint speed. It means if speed change incremental value is bigger than P111-P115, there is acceleration or deceleration. The purpose is to enhance connection of multi axis interpolation.

For example, if P111=300, during multi axis interpolation, if X speed changes from F800 to F1600, its process should be X axis changes to F1100 firstly, then accelerate to F1600.

- ★、 200,G00 continue run is valid[1 is No,16 is Yes]
- ★、 210,Thread is waiting for the spindle speed[0:No,1:Yes]
- ★、 211,Initial speed of rigid tapping spindle (rpm)
- ★、 212,The initial speed of the spindle with rigid tapping (rpm)
- ★、 213,Static tapping feed spindle acceleration and deceleration (rpm/S)[>1]
- ★、 214,Acceleration and deceleration of rigid tapping and cutting tool Spindle(rpm/S)[>1]
- ★、 215,Rigid tapping feed spindle reserved amount (1/1000 laps)[>2]
- ★、 216,Rigid tapping tool Spindle tail amount(1/1000 laps)[>2]

- ★、 217, The retention of rigid tapping and returning tool Spindle(1/1000 laps)[>2]
- ★、 218, Rigid tapping and retracting spindle trailing quantity(1/1000 laps)[>2]
- ★、 219, Time constant for smooth processing of rigid tapping [20002-20500]
- ★、 230, SP G00 speed(0.1rpm)
- ★、 231, SP G01 mode[+4:F,+8:G90/G91,+16:pulse]
 - +4 means execute SP command at the speed of F value(degree/min) to do orientation(at spindle orientation speed under G00), otherwise execute at the spindle orientation speed.
 - +8 means SP command is under control of G90/G91, under G90 means absolute reduction, G90 means incremental and not reduction, otherwise SP do reduction and run to nearest direction.
 - +16 means SP value display will be defined by sending pulse, otherwise defined by encoder feedback.
 - +8192 means spindle analog will be sent repeatedly per second.
 - +16384 means after executing M05, both spindle and analog output stop.
- ★、 232, SP direction[0:forward,1:negative,others:nearest]
- ★、 233, Control mode of spindle back to zero(1:pulse,2:servo control:out-M61,in-M22)
 - When set P233=16 or 23, firstly, spindle will stop, then controller send M75 to detect signal M22 from spindle driver(if P233=32, do not detect M22), then controller will calculate spindle current position, then position the degree P235 by the speed of P234. if offset is big, then set Axis parameter P56=16 to reverse feedback direction.
- ★、 234, SP oriented stop angle(0.1degree)
- ★、 235, SP home speed(0.1 rpm)

3.7.3 Axis parameter

- ★ 1, Feed axis band switch[1 mean Yes, 0 mean No]
 - Used to choose the feed speed ratio switch, 1 means choose external band switch, 0 means use “Feed ↑” and “Feed ↓” button.
 - Note: when P1=1, external band switch need to connect CN11 port- VDK0 (OFF), VDK1(*100), VDK2(*10), VDK3(*1) signal.
- ★ 2, Spindle and G00 band switch[1 mean Spindle, 2, mean G00, 0 mean No]
 - Used to choose the spindle speed ratio switch, 1 means choose external band switch, 0 means use “SP ↑” and “SP ↓” button.
 - Note: when P1=1, external band switch need to connect CN11 port- VDS0 (HCOM), VDS1(Z), VDS2(Y), VDS3(X) signal.
- ★ 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)

The movement scope of XYZ in machine coordinate.

★ 7,Spindle stop time(10ms)

The time which spindle need to brake, namely output time of M05, the shorter brake time means spindle stops faster.Unit:10ms.

★ 8,Spindle stop long signal

When P14=1, spindle brake signal is long signal, when P14=0, spindle brake time is short signal.

★ 9,Check SP encode[1 mean Yes,0 mean No]

Used to set if check spindle encoder signal. 1 means detect, 0 means not detect.

The parameter is used to activate such function relative to spindle speed like display speed speed, feed vale. And condition is machine need to be installed with encoder which is connected with spindle by 1:1 transmission ratio.

★ 10,SP encode pulse(4 times encode pulse)

The pulse unit per turn of Spindle encoder, the parameter valuse=encoder resolution*4.

★ 10-1,SP encode pulse alarm (diagnostics)[>10:Valid]

The tolerant pulse unit of spindle encoder feedback. Effective when more than 10.

★ 10-2,SP encode pulse[4 times encode thread](When Encode:SP is not 1:1)

Four times SP encoder pulse, if the encoder is connected with spindle by 1:1, set parameter as 0, otherwise, the valuse should be integrator bigger than 99.

★ 11,Soft limit invalid[D2X;D3Y;D4Z;D5A;D6B;D9C;D10XS;D11YS;1:No limit]

D2 means X axis;D3 means Y axis;D4 means Z axis;D5 means A axis;D6 means B axis;D9 means C axis;D10 means Xs axis;D11 means Ys axis;1 means not limit;

Set each axis in each bit. 1 means invalid soft limit in corresponding bit, 0 means valid.

Format: D15D14.....D1D0.

Example: If set soft limit of X axis, then set as:00000100

★ 12,X-axis's reverse compensation(um)

★ 13,Z-axis's reverse compensation(um)

If there is backlash in transmission structure when the axis move backward, it is necessary to set backlash compensation. P12-P13 backlash compensation of XZ. Unit:um.

★ 14,X-axis's direction signal[1 mean normal,0 mean reverse]

★ 15,Z-axis's direction signal[1 mean normal,0 mean reverse]

P14-P15 are used to set moving direction of axis, 0 means moving direction

reverse from command movement, 1 means axis moving direction is the same as command.

- ★ 16, Close feed electron gear[1 mean Yes, 0 mean No]

Used to disable electronic gear ratio, if P26=1, it means close electronic gear, then P27-P36f will not work.

- ★ 17, X-axis's electron gear numerator(1-999999)
- ★ 18, X-axis's electron gear denominator(1-999999)
- ★ 19, Z-axis's electron gear numerator(1-999999)
- ★ 20, Z-axis's electron gear denominator(1-999999)

Used to set XZ axis electronic gear ratio.

The numerator=10*reduction ratio;

The denominator = ball screw pitch;

If above parameter setting incorrect, axis will move incorrect, the command distance may not match with real movement.

- ★ 21, XZ positive limit[0 open, 1 close]
- ★ 22, XZ negative limit[0 open, 1 close]

Used to set XYZA axis limit type, 0 means Normal open, 1 means Normal close.

- ★ 23, float zero bit parameter[D3X;D4Y;D5Z;D6A;D7B;D8C;D9XS;D10YS;1:float Zero;D0D1:Y reduction]

Used to set the type of each axis home type, and it is a bit parameter.

D10|D9|D8|D7|D6|D5|D4|D3|D2|D1|D0

D3-D10 corresponds to XYZABCXsYs home type, 1 mean float zero point, 0 means mechanical home switch;

D1|D0 set if Y axis coordinate reduction;

For example, if just set X axis as float home type, then P23=00000001000.

- ★ 24, X coor float zero set
- ★ 25, Z coor float zero set

P24-P25 will set the current coordinate of XZ axis as home position, each axis goes home means back here.

If without mechanical zero point, set it as floating zero point. System will locate the zero point rapidly in operation of back to zero point.

- ★ 26, Feed axis home[1 mean No use, 0 mean clew, 8 compulsion, 9 must compulsion]

Set as 0 means no need, system will not prompt and limit when power on.

Set as 1 means prompt, system will prompt but no limit when power on.

Set as 8 means force, system will prompt when power on, and if no operation of back reference point, it will refuse to execute program.

Set as 9 means super force, motion system of feeding axis will prompt when power on, and if no operation of back reference point, it will refuse to execute motion.

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

Few methods for setting detection switch and zero pulse signal of motor encoder when each axis is backing to zero point:

Set as 0: When backing to zero point, crush to the reference switch, running reversal after the switch is off, then detect the zero pulse signal of motor encoder.

Set as 1: When backing to zero point, crush to the reference switch, running reversal after the switch is off.

Set as 2: When backing to zero point, crush to the reference switch, continue to run after the switch is off, then detect the zero pulse signal of motor encoder.

Set as others: When backing to zero point, crush to the reference switch, continue to run after the switch is off.

According to the circuit to set, normally suggest to set as 0 or 2, if not to detect the zero pulse signal of motor encoder, the accuracy will be worse, it only base on switch of reference point.

- ★ 28, Home reverse direction[D2X;D3Y;D4Z;D5A; D6B;0 Positive;1 Negative]

Exclusive setting of direction and sequence for every axis.

D10|D9|D8|D7|D6|D5|D4|D3|D2|D1|D0

D2 control X home direction; D3 control Y home direction; D4 control Z home direction; D5 control A home direction; D6 control B home direction, 1 means negative, 0 means negative.

For example, if set X axis home direction is negative, then P47=000000100.

D1 is a status control bit of chuck, 1 means detecting, 0 means without detecting.

If D1=1, after chuck clamping ready M10, system will detect M12, only if M12 is valid, will system execute next step. After chuck unclamping ready M11, system will detect M14, only if M14 is valid, will system execute next step.

- ★ 29, Home switch set [D0X;D1Y;D2Z;D3A;D4B;1:close;0:open;

D6=1handwheel&auto; D7=1manual&auto]

Used to set home switch type[NC/NO].

D15|D14|D13|D12|D11|D10|D9|D8|D7|D6|D5|D4|D3|D2|D1|D0

1 means normal close, 0 means normal open. D0 control X home direction; D1 control Y home direction; D2 control Z home direction; D3 control A home direction; D4 control B home switch type; D5 control C home switch type; D14 control Xs home switch type; D15 control Ys home switch type.

For example, if XYZ home switch is NC, then P29=000000000000111.

If D6=1, controller will switch between handwheel and auto mode automatically.
If D7=1, controller switch between manual mode and auto mode automatically,
controller will switch into manual mode after program finish, pressing Start button in
manual mode will switch into auto mode automatically.

★ 30,X check zero max lenth(100um)

★ 31,Z check zero max lenth(100um)

When P27=0 or 2, after release from home switch, each axis need to detect
motor encoder zero pulse signal, P30-P31 are the detecting range.

Note: the range must be less than distance by one type of motor, otherwise,
home position accuracy can not be ensured.

★ 32,X Home offset(10um)

★ 33,Z Home offset(10um)

During axis homing, when detect mode zero pulse signal, and move offset
vale, then homing finished. Unit: 0.01mm, range: -99999~+99999.

The parameter value depends on installation of home switch and machine
coordinate setting.

Note: offset speed is G00 speed.

★ 50,Have Spindle class control[1:Yes,0:No,64:Hold the spindle state]

Used to set if activate spindle motor when spindle switches gears. 1 means
activate, 0 mean not activate, 64 means hold current status.

★ 51,Spindle class speed(1/100rpm)

The spindle initializing speed setting when spindle switches gear.

★ 52,Spindle class direction[0 mean M03,1 mean M04]

The spindle rotary direction when spindle switches gear, 0 means rotate CW,
1 means rotates CCW.

★ 53,Spindle class stop time(10ms)

Used to set the stop time(M05) before initialize spindle when spindle
switches gear.

★ 54,Spindle class time(10ms)

Used to set spindle initialization time of spindle when spindle switches gear.

★ 55,Spindle stop time(10ms)

The delay time between canceling M03/M04 and output M05.

★ 68,XYZAB reverse delay time(ms)

The delay time of feed axis when it moves reverse.

★ 80,XZ axis coordinate plan[D2:Z work piece,D3:X work piece,D4:Z tool,D5: X
tool,D6:Z circumrotate,D7:X circumrotate]

Used to set if X and Z is rotary axis and do coordinate plan. 1 means yes, 0

means no. The parameter is bit parameter.

D7|D6|D5|D4|D3|D2|D1|D0.

D2 control Z axis work piece coordinate plan; D3 control X axis work piece coordinate plan; D4 control Z axis machine coordinate plan; D5 control X axis machine coordinate plan; D6 control Z axis is rotary axis, D7 control X axis is rotary axis.

- ★ 101, Lathe third axis name setting[0Y,1C]
- ★ 102, Lathe c axis function setting[0:rotation axis, 1:linear axis]
- ★ 103, lathe C is circumrotate axis[0 null;1 absolute coordinate plan;2 tool coordinate plan;3 all]
- ★ 104, C(Y)-axis's direction signal[1 mean normal,0 mean reverse]
- ★ 105, C(Y)-axis's electron gear numerator(1-32767)
- ★ 106, C(Y)-axis's electron gear denominator(1-32767)
- ★ 107, C(Y)Home offset(10um)
- ★ 108, C(Y)-axis's G00 speed(mm/min)
- ★ 109, C(Y)G1G2G3 max speed(mm/min)
- ★ 110, C(Y) acceleration
- ★ 111, C(Y)Handwheel limit speed(mm/min)
- ★ 112, C go home rampit speed(mm/min)
- ★ 113, Y go home rampit speed(mm/min)
- ★ 114, Y go home reverse speed(mm/min)
- ★ 115, Y check zero max lenth(100um)
- ★ 116, Y Home offset(10um)
- ★ 117, C(Y) axis's negative scope(mm)
- ★ 118, C(Y) axis's positive scope(mm)
- ★ 119, C(Y) coor float zero set
- ★ 200, A-axis function[0 mean circumrotate axis,1 mean line axis]
- ★ 202, A-axis is circumrotate axis[0 null;1 absolute coordinate plan;2 tool coordinate plan;3 all]
- ★ 203, A-axis's direction signal[1 mean normal,0 mean reverse]
- ★ 204, A-axis's electron gear numerator(1-999999)
- ★ 205, A-axis's electron gear denominator(1-999999)
- ★ 206, A Home offset(10um)
- ★ 207, A-axis's G00 speed(mm/min)
- ★ 208, A G1G2G3 max speed(mm/min)
- ★ 209, A acceleration
- ★ 210, Handwheel A limit speed(mm/min)

- ★ 211,A go home rampit speed(mm/min)
- ★ 212,A go home reverse speed(mm/min)
- ★ 213,A check zero max lenth(100um)
- ★ 214,A Home offset(10um)
- ★ 215,A axis`s negative scope(mm)
- ★ 216,A axis`s positive scope(mm)
- ★ 217,A coor float zero set
- ★ 218,A-axis automatic release of output point or auxiliary relay
[10000+Y_No.20000+M_No.]
- ★ 219,A-axis automatic release of input point or auxiliary relay
[10000+X_No.20000+M_No.30000+Delay time(ms)]
- ★ 301,B-axis axis [0 mean circumrotate axis,1 mean line axis]
- ★ 302,B-axis is circumrotate axis
- ★ [0 null;1 absolute coordinate plan;2 tool coordinate plan;3 all]
- ★ 303,B-axis named C(0 no,1 yes)
Display B axis as B or C, 0 means display as B, 1 means display as C.
- ★ 304,B-axis's direction signal[1 mean normal,0 mean reverse]
- ★ 305,B-axis's electron gear numerator(1-999999)
- ★ 306,B-axis's electron gear denominator(1-999999)
- ★ 307,B-axis`s reverse compensation(um)
- ★ 308,B-axis's G00 speed(mm/min)
- ★ 309,B G1G2G3 max speed(mm/min)
- ★ 310,B acceleration
- ★ 311,Handwheel B limit speed(mm/min)
- ★ 312,B go home rampit speed(mm/min)
- ★ 313,B go home reverse speed(mm/min)
- ★ 314,B check zero max lenth(100um)
- ★ 315,B Home offset(10um)
- ★ 316,B axis`s negative scope(mm)
- ★ 317,B axis`s positive scope(mm)
- ★ 318,B coor float zero set
- ★ 323,Does the axis of rotation not move at the nearest distance
(+4:X;+8:Y/C;+16:Z ;+32:A;+64:B)
- ★ 400,Is the measurement of spindle position feedback (1:Yes,0:No)
Used to set if detect spindle position feedback.
- ★ 404,SP motor direction(0 reverse,1 normal)

- ★ 405,SP-axis's electron gear(0 Yes,1 No)
Used to set if activate spindle electronic gear ratio.
- ★ 406,SP-axis's electron low gear numerator(1-999999)
- ★ 407,SP-axis's electron low gear denominator(1-999999)
- ★ 408,SP-axis's electron high gear numerator(1-999999)
- ★ 409,SP-axis's electron high gear denominator(1-999999)
- ★ 410,Z-axis Interpolation tap SP name[91X,92Y,93Z,94A,95B,96C,97Xs,98Ys]
- ★ 411,Interpolation tap mode[2 follow encode;3 interpolation to SP]
- ★ 412,SP tooth number(<P213)
- ★ 413,,Encode number(>P212)
Encoder gear tooth number.
- ★ 414,ABC-axis is moving by (7/8/9,17/18/19,27/28/29/D5..D13=1/A by X/Y/Z,C by X/Y/Z,B by X/Y/Z)
Set ABC axis following XYZ.

3.7.4 Tool parameter

- ★ 1,Electrical tool function[1 mean Yes and not detect the signal of tool number,8 mean yes and detect the signal of tool number ,0 mean No]
Used to set if enable electrical turret, 0 means without electrical turret; 1 means with electrical turret, but not detect tool position signal; 8 means dtetct tool position signal after change tool ready.
Note:
1.if turret is linear tool, P1=0;
2.press C in tool compensation interface, then input tool quantity amount.
- ★ 2,Electrical tool number +1
Used to set tool quantity number+1, for example, if turret has 4 tools, then P2=5.
- ★ 3,Lathe type[0 mean forward and horizontal, 1 mean back and horizontal, 8 mean forward and vertical, 9 mean back and vertical]
Used to set lathe structure.
- ★ 4,Tool positive rotate max-time(s)
Used to set max time of turret rotating. If can not detect tool position successfully over the time, controller will stop turret and alarm. Unit:S.
- ★ 5,Delay time after tool positive rotate(ms)
Set the delay time after turret rotating CW to detect tool position signal. Unit:ms.
- ★ 6,Delay time after tool stop(ms)
Set the delay time after turret stops, namely the delay time between cancel T+ signal and output T- signal. Unit:ms.

★ 7,Tighten time of tool reverse rotate(ms)

Set time of turret roating CCW to clock turret, namely time of outputing T- signal.

Unit:ms.

★ 9,Signal of tool reach to position(1 mean yes, 0 mean no)

Used to set if detect lock signal (TOK) after turret rotating CCW ready.

★ 10,C Tool radius compensation's establish(0 mean A,1 mean B)

★ 11,C Tool radius compensation's cancel(0 mean A,1 mean B)

★ 20,Electrical tool mode[1 mean normal,0 mean coding tool]

Used to set turret control mode, when P20=1, controller runs default tool change process. When P20=0, controller will run macro program ProgramTool to change tool.

★ 32,Signal of tool position/WAT signal filtration[+256+512+1024 correspond to 2/4/8ms of signal of tool position, +2048+4096+8192 correspond to WAT signal 2/4/8 ms]

D+number means the bit from right to left:

D8=1 means filter detection signal of tool position by 2ms;

D9=1 means filter detection signal of tool position by 4ms;

D10=1 means filter detection signal of tool position by 8ms, D8-D10 can be 1 at the same time;

D11=1 means filter WAT waiting input signal by 2ms;

D12=1 means filter WAT waiting input signal by 4ms;

D13=1 means filter WAT waiting input signal by 8ms, D11-D13 can be 1 at the same time;

3.7.5 Other parameter

★、1,Set sub-panel type[1 hand hold,0 panel]

Used to set if installed with hand wheel, 1 means handhold, 0 means panel.

★、2,lather outside chuck[1 extroversion,0 diffidence]

Used to set lathe chuck control mode. When machine installs auto chuck, this parameter can be used to switch chuck control mode. Inward chuck: when M10 effective, chuck goes inward; Outward chuck: when M10 effective, chuck goes outward.

1 means inward chuck, 0 means outward chuck.

★、3,use control switch

If controller is 1000 series, used to set if control switch works, 88 means activated both in manual and auto mode, 99 means activated in auto mode, other values means close.

★、4,Have auto lubricate(0 yes/1 no)

Used to set if machine installed with automatic lubricate system.

★、5,Auto lubricate time(0.01s)

Used to set auto lubricate time, namely M32 hold time, unit:10ms.

★、6,Auto lubricate stop time(0.01s)

Used to set interval time between each lubrication, namely interval time between M32 valid. Unit: s.

★、 7,Door switch checking(0 no,1 yes)

Used to set if detect door switch signal, if set as 0, means without door switch; if set as 1, means with door switch. It is suggest to detect door switch for safety sake.

Note: 1. door switch is detected through M12;

2. when M12 is valid, in manual mode, feed axis coordinate can move, but in auto mode, system will pause program automatically.

★、 8,Door switch(0 open,1 close)

Type of door switch signal, 0 means normally open, 1 means normally closed.

★、 9,bit parameter

D1=1 clear to 0; D2=1 space ;D5=0 close spindle; D6 speed; D8=1 save M10 when power on;D10=1 arrange; D12=1 skip is invalid; D13=1 back to zero is invalid

The format: D15|D14|D13|D12|D11|D10|D9|D8|D7|D6|D5|D4|D3|D2|D1|D0.

D0 bit: Default as 1, can't change.

D1 bit: Set as 1 means the number of workpiece clear automatically when power on, set as 0 means keep.

D2 bit: Set as 1 means insert space into letters automatically when editing the program, set as 0 means not insert.

D3 bit: Default as 0, can't change.

D4 bit: Default as 0, can't change.

D5 bit: Set as 1 means not stop spindle rotation and cooling when pressing RESET, 0 means stop, default is 0.

D6 bit: Set as 1 means each axis work according to own speed, work with nonlinear trajectory when executing G00, 0 means simultaneous, reach at the same time. Default is 1.

D7 bit: Default as 0, can't change.

D8 bit: Set as 1 means save the status M10/M11 of spindle loose or tight when power off, recover this status when power on again. Set as 0 means spindle will be tight automatically when power on.Default is 1.

D9 bit: Default as 1, can't change.

D10 bit: Set as 1 means the number of line is arranged automatically when programming.

D11 bit: Set as 1, the output analog of the first spindle is the same time from the first, second spindle channel, the function of the second spindle is valid.

D12 bit: Set as 1, shield the skip function, “/” in front of the segment means invalid.

D13 bit: Default as 0, can't change.

D14 bit: Default as 0, can't change.

D15 bit: Set as 1, the tool set show the relative value, otherwise it shows the absolute value.

Pay attention: This parameter includes the bit which can't be changed. If change this bit, it will cause some problems with controller.

★、 10,Auto count part[1 mean Yes,0 mean No]

Used to count processed work piece quantity.

★、 11,Program edit number increase

if user want to count program lines automatically, just need to set P11=1.

★、 13,Interlock between Spindle & chuck(0 mean no)

Used to set if interlock between spindle and chuck.

0 means no relationship between spindle and chuck status;

1 means only if spindle is in M05 status, can chuck clamp or unclamp;

8 means only if spindle is in M05 status and spindle encoder keep still, can chuck clamp or unclamp;

16 means detect speed and interlock;

32 means when program running in auto mode and status is not pause, button of clamp can not work;

56 means super interlock. when program running in auto mode and status is not pause, button of clamp can not work;

The parameter setting depends on machine configuration and user preference, it is suggested to set as interlock in safety sake.

★、 14,Is available keys of lub&cool as running[1:Yes,0:No]

Used to set if coolant button works in auto mode. The parameter setting depends on user preference, when set it as 1, operator can control coolant system at any time, to meet processing needs.

★、 17,Servo alarm ALM/X20(0:open,1:close)

Used to set type of XYZA alarm, 0 means normally open, 1 means normal close.

★、 18,Spindle Alarm ALM1/X23(0:open,1:close)

Used to set type of alarm ALM2, 0 means normally open, 1 means normal close.

★、 19,Machine Alarm ALM2/X24 (0 open,1 close)

Used to set type of ALM3 alarm, 0 means normally open, 1 means normal close.

★、 20,Chuck control signal(0 single,1 doubleM10/M71)

Used to set chuck control signal quantity, if one-way valve, it needs 1 signal, if with two-way valve, it needs 2 signal.

0: means only one output signal M10 controls chuck clamp or unclamp, when M10 is effective, system executes clamping, when M10 is ineffective, system executes unclamping;

1: means M71 and M10 signal control chuck unclamping and clamping separately. When M10 is effective and M71 is ineffective, chuck clamps; when M10 is ineffective, and M71 is effective, chuck unclamps. Namely M10 output M10, M11 output M71. Axis parameter P40-D1 bit is used to set if check clamp/unclamp ready, 0 means without detection; 1 means detection, then system will detect if M14 is effective after outputting M10, detect if M12 is effective after outputting M11, only if M14/M12 is effective, will system continue next step.

★、 22,Outside chuck control(0 no,1 yesM16)

Used to set if there is an external switch controlling chuck clamping or unclamping, it is a loop signal, namely chuck status changes everytime when Signal becomes effective once(effective once, then unclamp; effective another 1 time, then clamp). when set as 0, means without external chuck control switch; set as 1, means

with external chuck control, the signal is M16.

★、24,M10M11 short signal time(s)

Used to set the hold time of M10,M71 when they are the short signal. If the value is 0, it means M10 and M71 is long signal. Unit:S(second)

★、25,Whether to configure an integrated drive[1:Yes,0:No]

Used to set if connect with NEWKer integrated type driver. If yes, then when driver works normal, controller will output Y05 to control motor brake.

★、26,Emerge Stop(0 open,1 close)

Used to set type of emergency stop button, 0 means normally open, 1 means normally closed. The default setting is normally closed.

★、27,Emerge Stop2(0 open,1 close)

Used to set the type of external emergency stop signal. 0 means normally open, 1 means normally closed.

★、28,Run status output(0 invalid,1 valid Y04 run,Y02 stop)

Used to set if controller output signal for status displaying. 0 means do not output, 1 means output, namely when machine works, if program running, then output Y04, if program stopped, then output Y02.

★、29,Alarm status output(0 invalid,1 valid Y03)

Used to set if output alarm signal, namely when machine is in alarm status, if P29=0, controller will not output alssrm signal, if P29=1, controller will output alarm signal Y03.

★、30,Set language(1 表中文, 0 mean English)

★、31,Is enable PLC program

★、32,Is enable High PLC program

Used to set system inner PMC operating. During running machine, it must be enabled.

★、35,soft-limit without home as manual[1 Yes,0 No]

Used to set if soft limit works before the axis go home.

★、36,Set system time[year-month-day-hour-minute]

Used to set system time, format: year-month-day-hour-minute.

★、37,Velocit of RS232 [0=7200; 1=9600; 2=14400; 3=19200; 4=38400; 5=57600; 6=115200]

Used to set baud rate in RS232 communication.

★、37-1,Serial port ModBus station number of OPC function [odd check 10000+,even check 20000+,no check 30000+]

The station number set in RS232, if controller station number is 2, and check mode is odd check, then set as 10002.

★、38,Lock Manual rampit func key[8 Yes]

★、39,Special parameter

★、40,Special parameter

★、41,Bake current parameter

Backup current parameter setting as default parameter. After setting machine ready, press P41 twice then set current parameter as default setting for future maintenance.

★、42,Resume original parameter

Restore the default parameter setting which is defined by pressing P41 last time.

★、50,Run from middle Program ask going last line point[8:Yes,0:No]

Used to choose if position at end of last section before running program in middle line.

★、120,Manual axis moving keying reverse(4:is X;8:is Y;16:is Z;32:is A)

Used to exchange the direction of XYZA button.

★、200,X axis feedback alarm error(pulse)[>1]

★、201,Y axis feedback alarm error(pulse)[>1]

★、202,Z axis feedback alarm error(pulse)[>1]

★、203,4 axis feedback alarm error(pulse)[>1]

★、204,5 axis feedback alarm error(pulse)[>1]

When controller activated position feedback function, if tacking error during axis running is bigger than P200-P204, system will alarm.

Bigger than 1 is valid. Press “G” to clear the feedback position ,clear position and the alarm.

★、205,X axis stop feedback alarm error(pulse)[>1]

★、206,Y axis stop feedback alarm error(pulse)[>1]

★、207,Z axis stop feedback alarm error(pulse)[>1]

★、208,4 axis stop feedback alarm error(pulse)[>1]

★、209,5 axis stop feedback alarm error(pulse)[>1]

When controller activated position feedback function, if tacking error after axis stopping is bigger than P205-P209, system will alarm.

Bigger than 1 is valid. Press “G” to clear the feedback position ,clear position, and the alarm.

★、210,X-axis's electron gear numerator[auto count: L screw(um)M encode pulse]

★、211,Y-axis's electron gear numerator[auto count: L screw(um)M encode pulse]

★、212,Z-axis's electron gear numerator[auto count: L screw(um)M encode pulse]

★、213,4-axis's electron gear numerator[auto count: L screw(um)M encode pulse]

★、214,5-axis's electron gear numerator[auto count: L screw(um)M encode pulse]

If controller connect with feedback encoder AB signal from driver, set P210-P214=10000.

★、215,X-axis's electron gear denominator[auto count: L screw(um)M encode pulse]

★、216,Y-axis's electron gear denominator[auto count: L screw(um)M encode pulse]

★、217,Z-axis's electron gear denominator[auto count: L screw(um)M encode pulse]

★、218,4-axis's electron gear denominator[auto count: L screw(um)M encode pulse]

★、219,5-axis's electron gear denominator[auto count: L screw(um)M encode pulse]

Default as 10000, please do not change it.

★、300,absolute encoder servo motor

[X-D2,Y-D3,Z-D4,A-D5,B-D6,C-D7,XS-D8,YS-D9,0 mean No,1 mean Yes]

If the servo motor is with absolute encoder, it is necessary to set P300 accordingly.

Format: D15|D14|D13|D12|D11|D10|D9|D8|D7|D6|D5|D4|D3|D2|D1|D0

0 means without absolute encoder, 1 means with absolute encoder motor. D2 control X axis; D3 control Y axis; D4 control Z axis; D5 control A axis; D6 control B axis; D7 control C axis; D8 control Xs axis; D9 control Ys axis.

- ★、301,absolute encoder address of lower 16bits multi-turn data
- ★、302,absolute encoder address of higher 16bits one-revolution data
- ★、303,absolute encoder address of lower 16bits one-revolution data

P301=low 16-bit communication address of multiturn absolute in driver;

P302=high 16-bit communication address of single turn absolute in driver;

P303=low 16-bit communication address of single turn absolute in driver;

Attention: Controller communicate with Servo driver through RS485, protocol is as following:

1) Drive station number: 1 means X axis, 2 means Y axis, 3 means Z axis, 4 means A axis, 5 means B axis, 6 means C axis, 7 means Xs axis, 8 means Ys axis.

2) RS485 baud rate is 19200, 8 data bit, 1 stop bit, RTU format, Odd check mode.

- ★、304,X-axis one-revolution pulse
- ★、305,Y-axis one-revolution pulse
- ★、306,Z-axis one-revolution pulse
- ★、307,4-axis one-revolution pulse
- ★、308,5-axis one-revolution pulse

P304-P308 are used to set resolution of motor absolute encoder, for example, if motor encoder resolution is 17bit, then $P304-P308c=131072$; if motor encoder resolution is 23bit, then $P304-P308c=8388608$.

- ★、309,X-axis one-revolution coordinate(nm)
- ★、310,Y-axis one-revolution coordinate(nm)
- ★、311,Z-axis one-revolution coordinate(nm)
- ★、312,4-axis one-revolution coordinate(nm)
- ★、313,5-axis one-revolution coordinate(nm)

When the machine installed with absolute encoder motor, P309-P313 are used to set coordinate when motor rotates one revolution, unit is nm. Negative value means reverse direction of coordinate counting.

One-revolution coordinate=(Axis parameter P18/P17) $\times 10^7$

For example, if X axis reduction ratio is 4:1, ballscrew pitch is 5, then X electronic gear ratio will be 8:1, then $P309=(1/8)*10000000=2500000$.

- ★、314,X-axis multi-turn coordinate offset[Input E to clear,EV to clear&Set]
- ★、315,Y-axis multi-turn coordinate offset[Input E to clear,EV to clear&Set]
- ★、316,Z-axis multi-turn coordinate offset[Input E to clear,EV to clear&Set]
- ★、317,4-axis multi-turn coordinate offset[Input E to clear,EV to clear&Set]
- ★、318,5-axis multi-turn coordinate offset[Input E to clear,EV to clear&Set]

When the machine installed with absolute encoder motor, P314-P18 are used to set motor absolute muti-turn offset value, input E will clear absolute multi-turn data, normally it is used to set home position or solve data flow when multi-turn value is too big.

- ★、319,Servo driver current/speed/power/load rate(284/283/435/231)16 bits address.

- ★、320,X-axis one-revolution coordinate denominator
- ★、321,Y-axis one-revolution coordinate denominator
- ★、322,Z-axis one-revolution coordinate denominator
- ★、323,4-axis one-revolution coordinate denominator
- ★、324,5-axis one-revolution coordinate denominator

When the machine is installed with absolute encoder motor, P320-P324 are used to set one-revolution coordinate denominator, normally, it is 1.

- ★、350,Internet[0 means close, 1 means open, 8 means open automatically]
- ★、351,IP address of gateway

Set as you want, it is better to keep first three numbers the same to address in LAN, for example: 192.168.1.1.

- ★、352,IP address

It is better to keep first three numbers the same to address in LAN, for example: 192.168.1.103

- ★、353,MAC address

Make sure every piece should be different address, for example:255:255:255:1

- ★、354,IP address of internet service

The IP address of computer, It is better to keep first three numbers the same to address in LAN, for example: 192.168.1.13

- ★、380,Automatic tool X setting initial coordinate(mm)
- ★、381,Automatic tool Y setting initial coordinate(mm)
- ★、382,Automatic tool Z setting initial coordinate(mm)
- ★、383,Automatic tool forward speed(mm/min)
- ★、384,Automatic tool slow forward speed(mm/min)
- ★、385,Automatic tool Z axis coordinate value(mm)
- ★、386,Automatic tool Speed of fast locating points(mm/min)
- ★、387,Automatic tool setting mode(1:Fixed tool,0:Floating tool)
- ★、388,Automatic tool Z axis minimum machine tool coordinate value(mm)
- ★、389,Automatic tool Z axis difference value(mm)

Define macro variables of the automatic tool setting gauge function are as follows (corresponding to the other parameters P380 - P389):

#380: The X axis machine coordinate of initial position when automatic tool setting;(Unit:mm)

#381: The Y axis machine coordinate of initial position when automatic tool setting;(unit:mm)

#382: The Z axis machine coordinate of initial position and returning point with automatic tool setting;(unit:mm)

#383: The negative speed of automatic tool setting;(mm/min)

#384: The positive speed of automatic tool setting;(mm/min)

#385: The Z axis coordinate of workpiece surface in current workpiece coordinate system after automatic tool setting;(mm)

#386: The speed which is rapid move to locating position with automatic tool setting;(mm/min)

#387: Automatic tool setting mode (1 means fixed point, 0 means floating point).

#388: The minimal machine coordinate value of Z axis (mm);

#389: The gap value of Z axis [The height which is the gauge surface relative to the workpiece surface(mm)];

Fixed point gauge means putting the gauge in a fixed position, every time the X Y Z axis are automatic running to the fixed point first in tool setting; But the floating point gauge search the tool setting gauge signal along negative of the Z axis.

The input point X25 is default to be the checking point of automatic tool setting gauge to input.

2.The instruction:

M880 (corresponding to ProgramUser0) automatic tool setting instruction;

M882 (corresponding to ProgramUser2),

M883 (corresponding to ProgramUser3) set the gap of Z axis.

3. Automatic tool setting steps:

1)Set the No.380--No.388 parameter in other parameter;

2)Set the No.389 parameter in other parameter to set the drop value of Z axis: this operation needs to be set only once.

A.Run M882 instruction in MDI to set the gap of Z axis;

B.Manual run Z axis to move the tool nose to the workpiece surface;

C.Run M883 instruction in MDI to automatic set the gap of Z axis No.389 parameter in other parameter;

3) MDI choose the workpiece coordinate system G54/G59;

4) Automatic tool setting: MDI running the M880 instruction, automatic set the Z axis offset of the current workpiece coordinate system.

★、400,Axis name display configuration [xxxxx2] (123456 express XYZABC, Example:123462 shows XYZAC,124502 shows XYAB)

★、401,Feed shaft pulse port configuration[xxxxxxxxx0] (12345678 express XYZABC78,Example:123478560 is 7/8/5/6 axis pulse output from B/C/XS/YS port)

★、451,Controlling mode of robot(100--9999)

★、452,Safety switch is work or not[1 work,0 not work]

★、453,The brake of motor is work or not[1 check the X40 when motor with brake, 0 without brake]

★、501,Interface display [1:is white,8:is black]

★、601,One key to set stepper type

Applies to case where the stepper equipment is installed in robot. To set the current parameters as parameter of factory default stepper motor. The parameter mainly adjusts the part parameters of speed and motor's specification.

★、602,One key to set servo type

Applies to case where the servo equipment is installed in robot. To set the current parameters as parameter of factory default servo motor. The parameter mainly adjusts the part parameters of speed and motor's specification.

★、900,User-defined dialog box[1:is invalid,4:is valid,12:is all]

Used to set if activate user-defined dialog when press Enter in main interface.

★、901,Axis go home sequence(>9) [5-9 bits,1/2/3/4/5/6/7/8 is X/Y/Z/A/B/C/XS/YS,

last bit must is 0]

Used to set sequence of each axis home. For example, if P901=4521360, it means axis go home sequence is 452136 axis.

★、903,Inner parameter[2start function of SD card,+16:B-axis Welding following processing function,+32:A-axis Welding following processing function]

★、910,M18/M22/M24/M28 High speed input[1:Yes]

Used to set if define M18/M22/M24/M28 as high speed input of G31/G311.

★、911,Whether to start M18 for the teaching, M28 for the recording function[1:Yes,0:No]

Used to set if activate teaching function by input point M18/M24/M28.

★、912,"Reset" to reset the output interface[1:Yes,0:No]

Used to set if reset output point when press Reset button.

★、994,CNC controller serial number

★、995,Machine tool serial number

★、996,user-defined serial number

3.7.6 Coordinate

The parameter has the function of multiple coordinate system, includes 6 workpiece coordinate system and a lathe coordinate system G53. A processing program can set a workpiece coordinate system, workpiece coordinate system could move its original to change,the value of coordinate system in parameter is coordinate value of the original point (zero point) in the lathe coordinate system.

Using G54 to G59 to set 6 workpiece coordinate system, in coordinate system interface could modify the coordinate value of original of 6 workpiece coordinate system in machine coordinate system.

For modification of G54-G59, input following instructions into dialog:

1) input Axxx means the certain axis of all coordinate offset, for example, if input

A12.5, it means current items(Axis) of all coordinate offset 12.5mm;

2) input E means clear, input Exxx means input absolute offset value;

3) input EA means the certain axis of all coordinate will be cleared. Input EAxxx means certain axis of all coordinate will be input the offset value.

4) in X axis of any coordinate system, input EALL means clear offset of all axis in all coordinate systems.

Bus type controller has 54 workpiece coordinate system: G54-G59, G54.1-G54.48.

1-0,Current set of workpiece coordinate[G54-G59]

1-1,X of work coordinates G54-G59(mm)

1-2,Y of work coordinates G54-G59(mm)

1-3,Z of work coordinates G54-G59(mm)

1-4,A of work coordinates G54-G59(mm)

1-5,B of work coordinates G54-G59(mm)

1-6,C of work coordinates G54-G59(mm)

1-7,Xs of work coordinates G54-G59(mm)

1-8,Ys of work coordinates G54-G59(mm)

2-0,Current set of workpiece coordinate[G54.1-G54.48]

2-1,X of work coordinatesG54.1-G54.48(mm)

2-2,Y of work coordinatesG54.1-G54.48(mm)
2-3,Z of work coordinatesG54.1-G54.48(mm)
2-4,A of work coordinatesG54.1-G54.48(mm)
2-5,B of work coordinatesG54.1-G54.48(mm)
2-6,C of work coordinatesG54.1-G54.48(mm)
2-7,Xs of work coordinatesG54.1-G54.48(mm)
2-8,Ys of work coordinatesG54.1-G54.48(mm)
1,X of work coordinates G54(mm)
2,Y of work coordinates G54(mm)
3,Z of work coordinates G54(mm)
4,A of work coordinates G54(mm)
5,B of work coordinates G54(mm)
6,C of work coordinates G54(mm)
7,Xs of work coordinates G54(mm)
8,Ys of work coordinates G54(mm)
11,X of work coordinates G55(mm)
12,Y of work coordinates G55(mm)
13,Z of work coordinates G55(mm)
14,A of work coordinates G55(mm)
15,B of work coordinates G55(mm)
16,C of work coordinates G55(mm)
17,Xs of work coordinates G55(mm)
18,Ys of work coordinates G55(mm)
21,X of work coordinates G56(mm)
22,Y of work coordinates G56(mm)
23,Z of work coordinates G56(mm)
24,A of work coordinates G56(mm)
25,B of work coordinates G56(mm)
26,C of work coordinates G56(mm)
27,Xs of work coordinates G56(mm)
28,Ys of work coordinates G56(mm)
31,X of work coordinates G57(mm)
32,Y of work coordinates G57(mm)
33,Z of work coordinates G57(mm)
34,A of work coordinates G57(mm)
35,B of work coordinates G57(mm)
36,C of work coordinates G57(mm)
37,Xs of work coordinates G57(mm)
38,Ys of work coordinates G57(mm)
41,X of work coordinates G58(mm)
42,Y of work coordinates G58(mm)
43,Z of work coordinates G58(mm)
44,A of work coordinates G58(mm)
45,B of work coordinates G58(mm)

46,C of work coordinates G58(mm)
47,Xs of work coordinates G58(mm)
48,Ys of work coordinates G58(mm)
51,X of work coordinates G59(mm)
52,Y of work coordinates G59(mm)
53,Z of work coordinates G59(mm)
54,A of work coordinates G59(mm)
55,B of work coordinates G59(mm)
56,C of work coordinates G59(mm)
57,Xs of work coordinates G59(mm)
58,Ys of work coordinates G59(mm)
61,X of work coordinates G54.1(mm)
62,Y of work coordinates G54.1(mm)
63,Z of work coordinates G54.1(mm)
64,A of work coordinates G54.1(mm)
65,B of work coordinates G54.1(mm)
66,C of work coordinates G54.1(mm)
67,Xs of work coordinates G54.1(mm)
68,Ys of work coordinates G54.1(mm)
71,X of work coordinates G54.2(mm)
72,Y of work coordinates G54.2(mm)
73,Z of work coordinates G54.2(mm)
74,A of work coordinates G54.2(mm)
75,B of work coordinates G54.2(mm)
76,C of work coordinates G54.2(mm)
77,Xs of work coordinates G54.2(mm)
78,Ys of work coordinates G54.2(mm)
81,X of work coordinates G54.3(mm)
82,Y of work coordinates G54.3(mm)
83,Z of work coordinates G54.3(mm)
84,A of work coordinates G54.3(mm)
85,B of work coordinates G54.3(mm)
86,C of work coordinates G54.3(mm)
87,Xs of work coordinates G54.3(mm)
88,Ys of work coordinates G54.3(mm)
91,X of work coordinates G54.4(mm)
92,Y of work coordinates G54.4(mm)
93,Z of work coordinates G54.4(mm)
94,A of work coordinates G54.4(mm)
95,B of work coordinates G54.4(mm)
96,C of work coordinates G54.4(mm)
97,Xs of work coordinates G54.4(mm)
98,Ys of work coordinates G54.4(mm)
101,X of work coordinates G54.5(mm)

102,Y of work coordinates G54.5(mm)
103,Z of work coordinates G54.5(mm)
104,A of work coordinates G54.5(mm)
105,B of work coordinates G54.5(mm)
106,C of work coordinates G54.5(mm)
107,Xs of work coordinates G54.5(mm)
108,Ys of work coordinates G54.5(mm)
111,X of work coordinates G54.6(mm)
112,Y of work coordinates G54.6(mm)
113,Z of work coordinates G54.6(mm)
114,A of work coordinates G54.6(mm)
115,B of work coordinates G54.6(mm)
116,C of work coordinates G54.6(mm)
117,Xs of work coordinates G54.6(mm)
118,Ys of work coordinates G54.6(mm)
121,X of work coordinates G54.7(mm)
122,Y of work coordinates G54.7(mm)
123,Z of work coordinates G54.7(mm)
124,A of work coordinates G54.7(mm)
125,B of work coordinates G54.7(mm)
126,C of work coordinates G54.7(mm)
127,Xs of work coordinates G54.7(mm)
128,Ys of work coordinates G54.7(mm)
131,X of work coordinates G54.8(mm)
132,Y of work coordinates G54.8(mm)
133,Z of work coordinates G54.8(mm)
134,A of work coordinates G54.8(mm)
135,B of work coordinates G54.8(mm)
136,C of work coordinates G54.8(mm)
137,Xs of work coordinates G54.8(mm)
138,Ys of work coordinates G54.8(mm)
141,X of work coordinates G54.9(mm)
142,Y of work coordinates G54.9(mm)
143,Z of work coordinates G54.9(mm)
144,A of work coordinates G54.9(mm)
145,B of work coordinates G54.9(mm)
146,C of work coordinates G54.9(mm)
147,Xs of work coordinates G54.9(mm)
148,Ys of work coordinates G54.9(mm)
151,X of work coordinates G54.10(mm)
152,Y of work coordinates G54.10(mm)
153,Z of work coordinates G54.10(mm)
154,A of work coordinates G54.10(mm)
155,B of work coordinates G54.10(mm)

- 156,C of work coordinates G54.10(mm)
- 157,Xs of work coordinates G54.10(mm)
- 158,Ys of work coordinates G54.10(mm)

3.7.7 Password

password setting includes:

- 1,Is enable CNC Co.'s password?(1 means No, 0 means Yes)
 - 2,Is enable Machine Co.'s password?(1 means No, 0 means Yes)
Original password is "NEWNEW".
 - 3,Is enable User's password?(1 means No, 0 means Yes)
Original password is "KERKER".
 - 4,Modify CNC Co.'s password:
 - 5,Modify Machine Co.'s password:
 - 6,Modify User's password:
 - 7,Curry Using time: (days)
 - 8.Print screen function
- Soft version.
Time

3.8 Set tool compensation

Press "Redeem" to enter interface of redeem in any interface, including "Radius compensation" "Redeem" "Clear all value" "Clear current value" "Measure tool" "Posit tool" "Set", total 7 functions, correspond to "F1-F7" to enter corresponding interface, press "Esc" to back the primary menu interface.

3.8.1 Radius compensation

Press "F1" enter radius compensation interface in redeem interface. The parameter is used to set adopt corner radius of the tool.

Setting method: Press "↑ ↓" to make cursor move to the corresponding tool and press "Enter" to pop up a dialog box, import corresponding tool radius, press "Enter" at last.

Press "F3" or "F4" to make all initial or current tools to be 0.

Pay 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.

Man Con		鑫科瑞数控 N00000		2019-03-19 14:57																																							
<table border="1"> <tr><td>01</td><td>X: 0.000</td><td>Z: 0.000</td><td>C: 0.000</td><td>A: 0.000</td></tr> <tr><td>02</td><td>X: 0.000</td><td>Z: 0.000</td><td>C: 0.000</td><td>A: 0.000</td></tr> <tr><td>03</td><td>X: 0.000</td><td>Z: 0.000</td><td>C: 0.000</td><td>A: 0.000</td></tr> <tr><td>04</td><td>X: 0.000</td><td>Z: 0.000</td><td>C: 0.000</td><td>A: 0.000</td></tr> <tr><td>05</td><td>X: 0.000</td><td>Z: 0.000</td><td>C: 0.000</td><td>A: 0.000</td></tr> <tr><td>06</td><td>X: 0.000</td><td>Z: 0.000</td><td>C: 0.000</td><td>A: 0.000</td></tr> <tr><td>07</td><td>X: 0.000</td><td>Z: 0.000</td><td>C: 0.000</td><td>A: 0.000</td></tr> <tr><td>08</td><td>X: 0.000</td><td>Z: 0.000</td><td>C: 0.000</td><td>A: 0.000</td></tr> </table>		01	X: 0.000	Z: 0.000	C: 0.000	A: 0.000	02	X: 0.000	Z: 0.000	C: 0.000	A: 0.000	03	X: 0.000	Z: 0.000	C: 0.000	A: 0.000	04	X: 0.000	Z: 0.000	C: 0.000	A: 0.000	05	X: 0.000	Z: 0.000	C: 0.000	A: 0.000	06	X: 0.000	Z: 0.000	C: 0.000	A: 0.000	07	X: 0.000	Z: 0.000	C: 0.000	A: 0.000	08	X: 0.000	Z: 0.000	C: 0.000	A: 0.000	Program 99.TXT Progress <div style="width: 100%; height: 10px; background-color: #ff00ff;"></div> G53 T0000 TrueFeed F 0.000 M05 M09 M10 M78 M33 M41 G00 X100% F2000 X150% S100 X100% SP000 Machine Coord X 164.430 A 0.000 Z 157.845 C 0.000 PartTime 0:0 PartNo 171 SPrpm 0 No Alarm	
01	X: 0.000	Z: 0.000	C: 0.000	A: 0.000																																							
02	X: 0.000	Z: 0.000	C: 0.000	A: 0.000																																							
03	X: 0.000	Z: 0.000	C: 0.000	A: 0.000																																							
04	X: 0.000	Z: 0.000	C: 0.000	A: 0.000																																							
05	X: 0.000	Z: 0.000	C: 0.000	A: 0.000																																							
06	X: 0.000	Z: 0.000	C: 0.000	A: 0.000																																							
07	X: 0.000	Z: 0.000	C: 0.000	A: 0.000																																							
08	X: 0.000	Z: 0.000	C: 0.000	A: 0.000																																							
F1 Radius F2 Length F3 ACLEA F4 CLEAR F5 SetTool F6 ToolPoint F7 Set F8 COOR																																											

3.8.2 Length compensation

Press “F2” to enter length of redeem interface. The parameter is used to modify the length which is adopt or reset the length.

Method of modifying the length:

Press “↑ ↓” to make cursor move to the corresponding tool number and press “Enter” to popup a dialog box, import the modifying axis into the dialog box and import the modifying value(import 0.05 to plus 0.05, import -0.05 to reduce 0.05), press “Enter” to confirm. The system calculates current value of redeem after finishing setting.

Method of reset the length:

Make lathe move to a position so that measure corresponding tool coordinate, press “↑ ↓” to make cursor move to corresponding tool number and press “F5” to popup a dialog box, import the reset axis into dialog box and import the value of measuring the work piece of corresponding axis, press “Enter” to confirm. The length compensation of corresponding axis has been reset. The system automatical refresh current value of redeem after finishing setting.

Method of initializing the length compensation value of tool:

Press “F3” or “F4” to initialize all the length compensation or current length compensation.

3.8.3 Posit tool

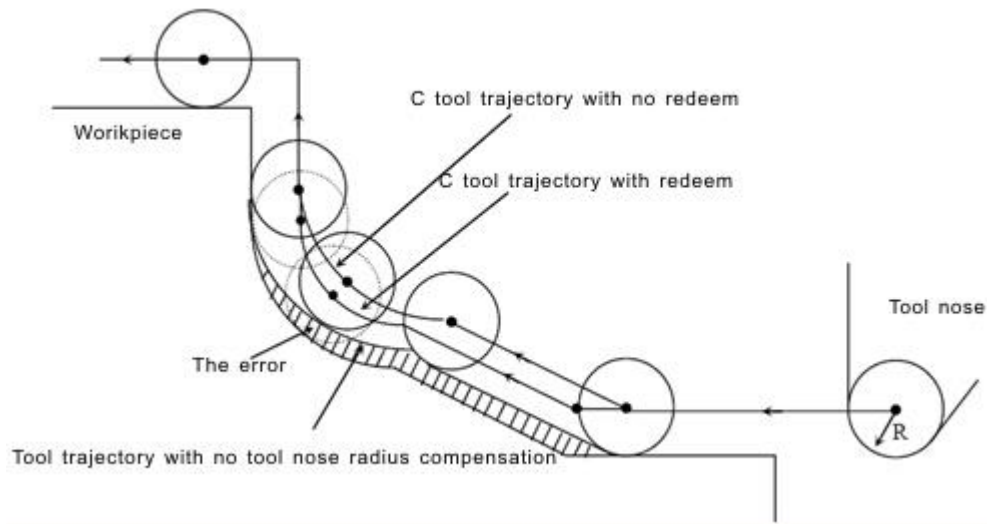
Press “F6” to enter posit tool interface in redeem interface. The parameter is used to set the kind of tool when adopting radius compensation of tool.

Method of setting: Press “↑ ↓” to make cursor move to corresponding tool number and press “Enter” to popup a dialog box, import the code of corresponding tool kinds and press “Enter” to confirm.

Press "F1" to initialize all the kinds of tools to 0.

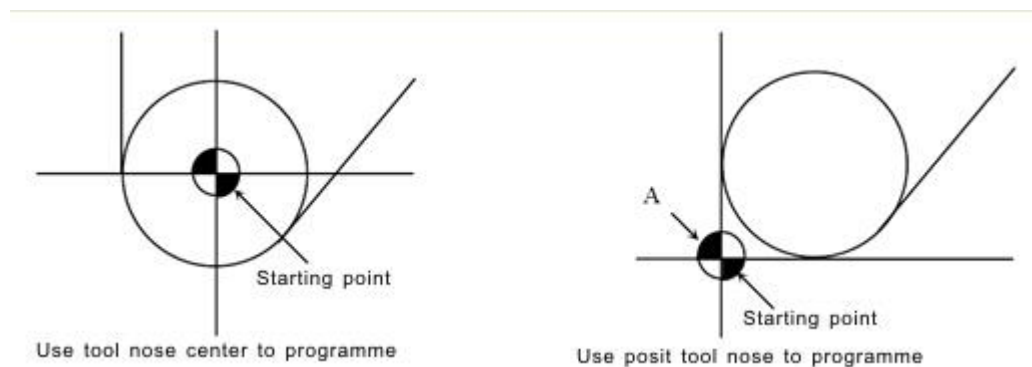
Introduction of posit tool

Only use the function of tool compensation offset when the tool is cycle to make a correct processing program is very difficult. The function of tool compensation could compensate automatically.



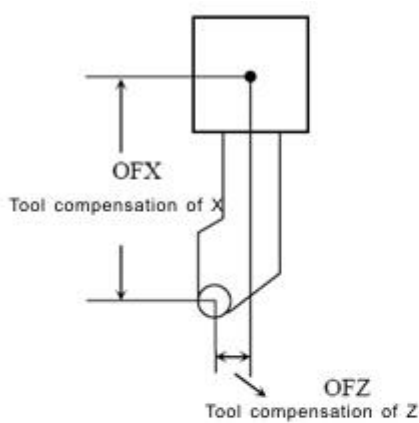
A. Posit tool

The nose of tool point(A) is non-existent actually as follow picture. Setting posit tool is easier than setting actual center of tool (Hereinafter). Using posit tool to programme do not need to consider the tool radius. The tool in the origin of the position relation as shown below.

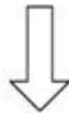


Pay attention: For some lathes with electrical home point, a standard point(like center nose of tool) can be the starting point. The distance form the standard point to the radius center of tool rest or posit tool can be the offset of tool.

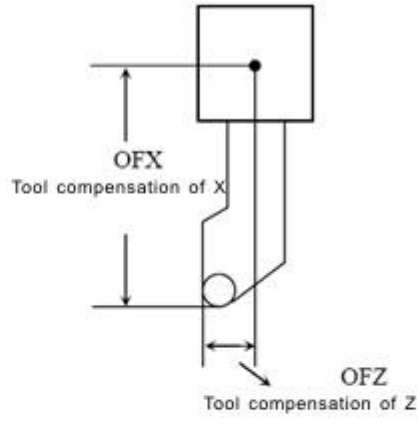
When tool rest is at the beginning point.



Set the distance from standard point to tool nose center as the compensation



Put the starting point on tool nose center



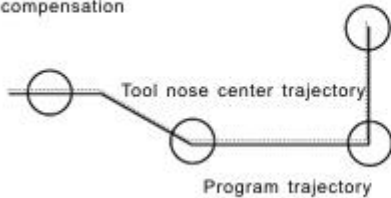
Set the distance from standard point to tool nose as the compensation



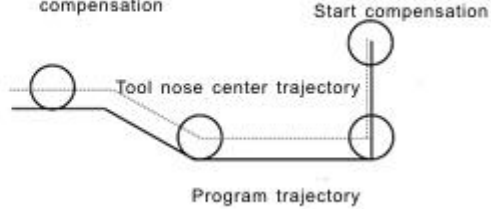
Put the starting point on posit tool nose

Using the central nose of nose to programme

Tool nose center trajectory is the same as program trajectory when there is no tool nose radius compensation



It will cut correctly if there is tool nose compensation

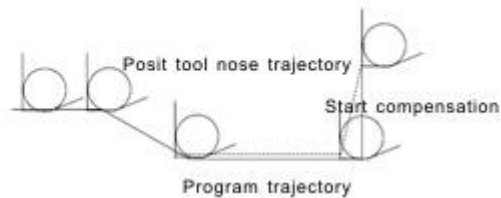


Using the posit tool to programme

Posit tool nose trajectory is the same as program trajectory when there is no tool nose radius compensation

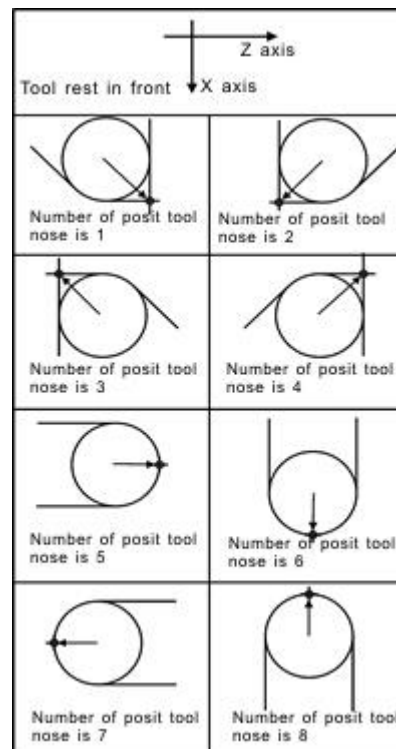
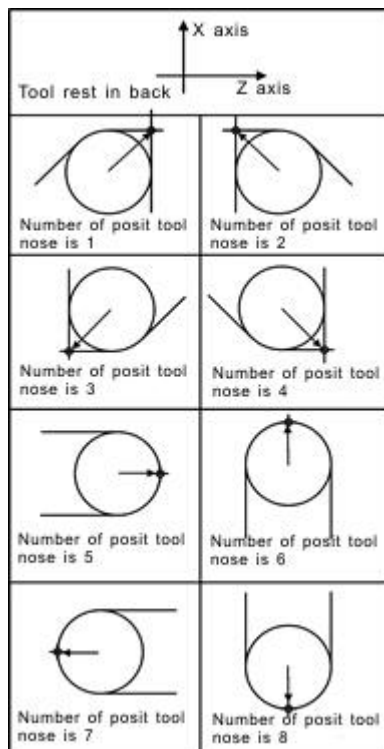


It will cut correctly if there is tool nose radius compensation



B. Direction of posit tool

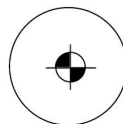
From the central nose of tool to see the direction of posit tool is determined by the direction of cutting tool, so must set before as the same time with the value of compensation. The direction of posit tool can be selected by the 8 kinds of corresponding number as shown below.



The direction of posit tool of backward tool rest

The direction of posit tool of forward tool rest

Using No.0 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 “F7” to popup a dialog box in the redeem interface to set and manage the total tools.

Including sum tools of electrical tool rest and toolpost.

3.9 Screw compensation

Press “Parameter” twice in parameter interface to enter screw compensation interface to set the screw compensation.

Screw compensation is used for automatical compensating the error of screw pitch, compensate the influence from the error of screw pitch to the prevision of operating lathe. The system adopts storage mode of screw compensation: Making the lathe’s home point as the starting point when debugging, measured the error curve of screw, studied out the correctional curve according to the error curve, import the value of correctional curve into the correctional parameter and system is going to compensate

according to the parameter in automatical running.

Man Con		N00000		2013-08-23 11:29	
Basic Parameter		No	Coord(mm)	Value(um)	
1, Standby: 0		1)	<320.000>	0	
2, Neg-point: 3		2)	<160.000>	0	
3, Pos-point: 2		3)	<0.000>	0	
4, multiple: 1.000		4)	<-160.000>	0	
5, distance(um): 80000		5)	<-320.000>	0	
		6)	<-480.000>	0	
		Program 99.TXT			
		Instruction code			
		G53			
		T0000			
		Machine Status			
		M05	M09	M10	
		M78	M33	M70	
		G00	X100%		
		F30	X100%		
		S50	X100%		
		Machine Coord			
		X	0.000 A	0.000	
		Z	0.000 B	0.000	
		C	0.000	SP360	
		PartTime	0: 0		
		PartNo	0		
		SPrpm	0		
No Alarm					
X-axis	Z-axis	C-axis	A-axis	B-axis	ChePro CLEAR CANCEL

Screw compensation interface

Screw compensation by the axis as the unit to set storage, set X Z axis separately, by pressing “F1” “F2” to switch; Every axis of screw compensation interface has tow areas(basic parameter and set the compensation), by pressing “→ ←”to move the cursor to realize.

Basic parameter:

Press “↑ ↓” to select current basic parameter to set in basic parameter, press “Enter” to popup a dialog box to import the error compensation of every axis and import the basic information of screw compensation.

Set compensation:

In the area of setting compensation, it will shows the value of compensation and every axis' error compensation point of screw pitch. Press “↑ ↓ PgDn PgUp” to select current compensation point and press “Enter” to popup a dialog box to import the value of compensation, import the value of current compensation point.

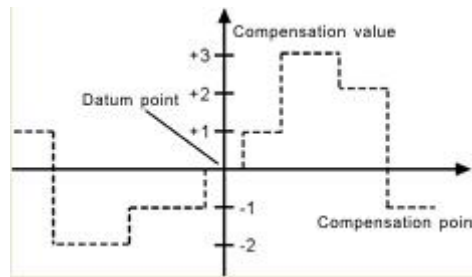
Test program generation automatically

Automatical generate a program of laser interferometer to check the screw compensation. Enter the screw pitch interface and set basic parameter, press checking program to popup a dialog box and press “Enter” to generate corresponding checking program of screw compensation.

The number of compensation points can be set freely, the maximum number of each axis is 300. 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 calculates every axis' error compensation points' positions of screw pitch according to basic parameter automatically, every axis's error compensation points' spacing is uniform, user can import compensation value of each point (This system requires importing absolute value, relating the of home point).

The compensation points are uniform, set the spacing into each axis.

For example:

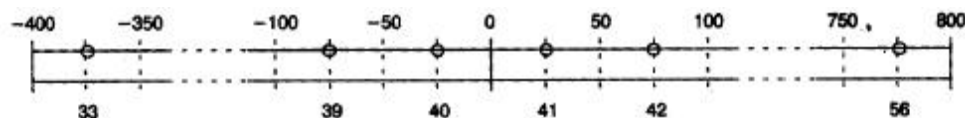
Example 1: Linear axis: when length of travel is -400mm~+800mm, spacing of points is 50mm, reference point compensation is No.40, it can figure out that Compensation point of farthest end in negative direction is:

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 value in 0 position

parameters 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

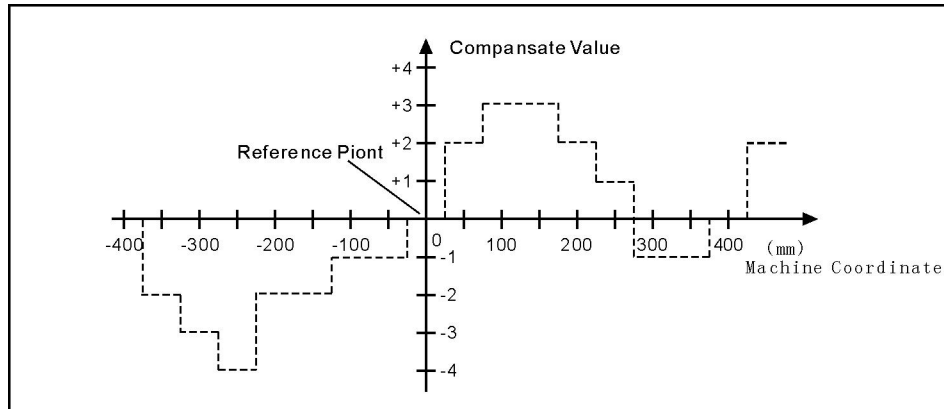
Corresponding compensation point and value:

The compensation value in corresponding compensation point:

No.	3	3	3	3	3	3	3	4	4	4	4	4	4	4	4	4	4	5
	3	4	5	6	7	8	9	0	1	2	3	4	5	6	7	8	9	6

Value	+	+	+	-2	0	-1	0	-1	+	+	0	-1	-1	-2	0	+	+	+
	2	1	1						2	1						1	2	1

The contrasted chart of compensation points and value as follows:

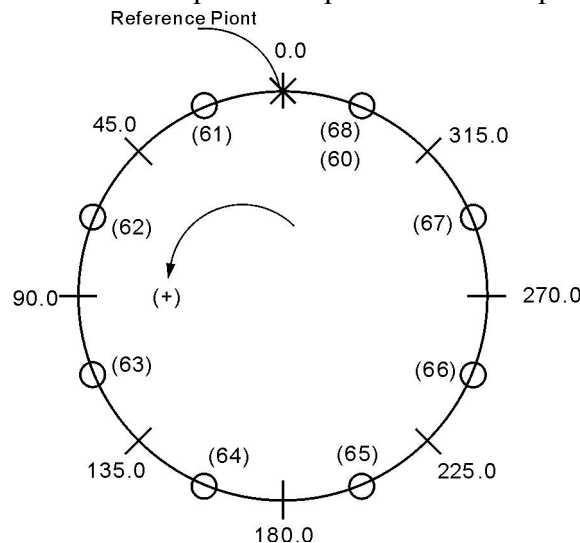


Example 2: rotor axis: when movement per revolution is 360°, interval of points 45°, reference point compensation NO. 60, Compensation point NO. of farthest end in negative direction is usually same as reference point compensation point NO.

Compensation point NO. of farthest end in positive direction is:

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

Machine coordinate and compensation point NO. correspondence is:



Note: input value in small circle. If the total amount from 61 to 68 doesn't equal 0, accumulated pitch error per revolution will deviate, so same value shall be put 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

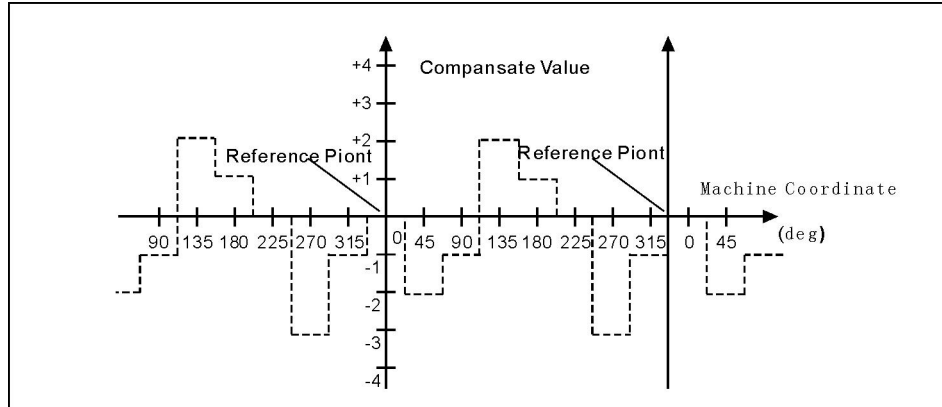
Compensation percentage: 1

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 in parameter interface.

System diagnosis interface(Input signal)

Press “F2” and “↑ ↓” to check the status of input and output, press “F4” to check alarm information.

Man Con		N00000		2013-02-26 15:05	
Input point					
X00 T01	X01 T02	X02 T03	X03 T04	X04 T05	X05 T06
X06 T07	X07 T08	X08 M34/A0	X09 -L	X10 +L	X11 M36/Y0
X12 X0	X13 Z0	X14 KRUN	X15 KHALT	X16 Z20	X17 Z20
X18 KLEFT	X19 KRIGHT	X20 STOP	X21 T0K	X22 ALM	X23 ALM1
X24 ALM2	X25 M28	X26 M24	X27 M22	X28 M18	X29 M12
X30 M14	X31 M16	X32 HX	X33 HY	X34 HZ	X35 HA
X36 HX1	X37 HX10	X38 HX100	X39 H0FF	X40	X41
X42	X43	X44	X45	X46	X47
X60 DS3	X61 DS2	X62 DS1	X63 DS0	X64 DK3	X65 DK2
X66 DK1	X67 DK0				
No Alarm					
Program %99.TXT					
Instruction code					
G53					
T01H0D0					
Machine Status					
M05		M09		M10	
M78		M33		M70	
G00		X100%			
F120		X100%			
S0		X100%			
Machine Coord					
X		0.000 A		0.000	
Y		0.000 B		0.000	
Z		0.000		SP360	
PartTime		0: 0			
PartNo		0			
SPrpm		0			
I/O					
ALARM					
Reset					
CANCEL					

Checking interface of output signal

In the interface of output or input, No.0 or No.1 stands for status, 1 means effective, 0 means no effect.

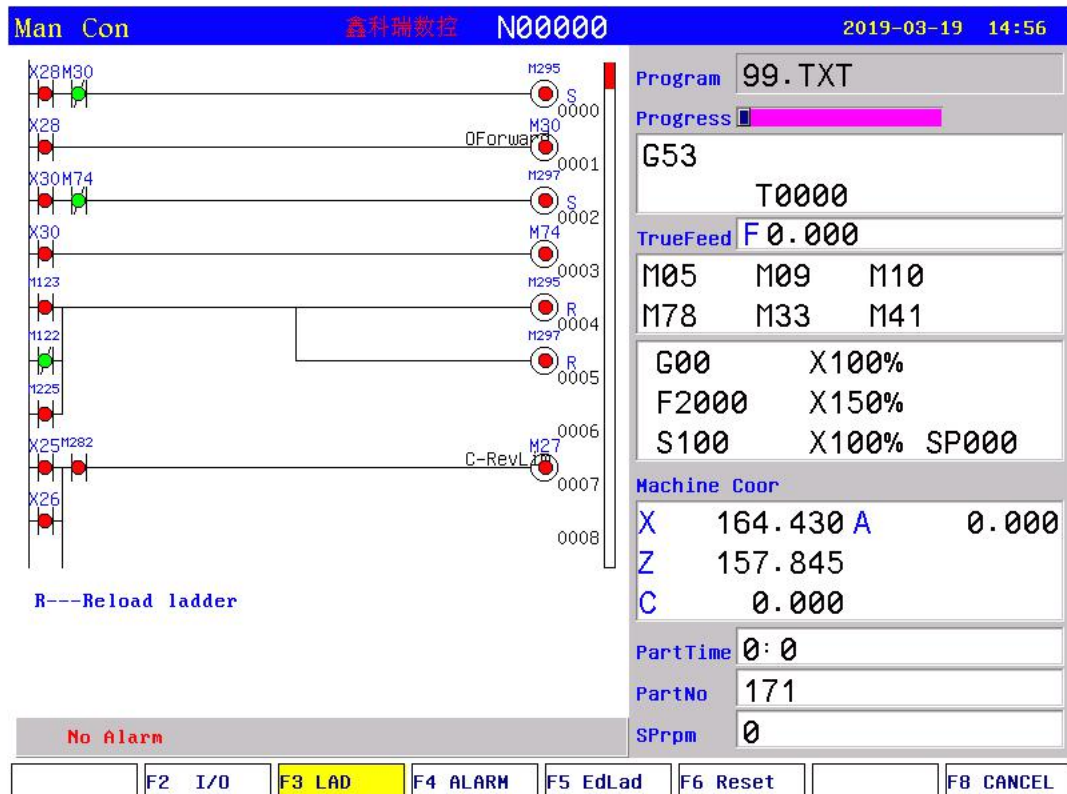
Man Con		N00000								2013-02-26 15:06																																																																																																															
<div>Output Point</div> <table border="1"> <tr> <td>0</td><td>0</td><td>0</td><td>0</td><td>0</td><td>0</td><td>0</td><td>0</td><td>0</td><td>0</td> </tr> <tr> <td>Y00</td><td>Y01</td><td>Y02</td><td>Y03</td><td>Y04</td><td>Y05</td><td>Y06</td><td>Y07</td><td>Y08</td><td>Y09</td> </tr> <tr> <td>M61</td><td>M63</td><td>M65</td><td>M67</td><td>M69</td><td>M71</td><td>M73</td><td>M75</td><td>M77</td><td>M79</td> </tr> <tr> <td>0</td><td>0</td><td>0</td><td>0</td><td>0</td><td>0</td><td>0</td><td>0</td><td>0</td><td>0</td> </tr> <tr> <td>Y08</td><td>Y09</td><td>Y10</td><td>Y11</td><td>Y12</td><td>Y13</td><td>Y14</td><td>Y15</td><td>Y16</td><td>Y17</td> </tr> <tr> <td>M32</td><td>M34</td><td>M36</td><td>M38</td><td>M40</td><td>M42</td><td>M44</td><td>M46</td><td>M48</td><td>M50</td> </tr> <tr> <td>0</td><td>0</td><td>0</td><td>0</td><td>0</td><td>0</td><td>0</td><td>0</td><td>0</td><td>0</td> </tr> <tr> <td>Y16</td><td>Y17</td><td>Y18</td><td>Y19</td><td>Y20</td><td>Y21</td><td>Y22</td><td>Y23</td><td>Y24</td><td>Y25</td> </tr> <tr> <td>LRUN</td><td>INTH</td><td>+T</td><td>-T</td><td>S04</td><td>S03</td><td>S02</td><td>S01</td><td>Y26</td><td>Y27</td> </tr> <tr> <td>0</td><td>0</td><td>0</td><td>0</td><td>0</td><td>0</td><td>0</td><td>0</td><td>0</td><td>0</td> </tr> <tr> <td>Y24</td><td>Y25</td><td>Y26</td><td>Y27</td><td>Y28</td><td>Y29</td><td>Y30</td><td>Y31</td><td></td><td></td> </tr> </table>										0	0	0	0	0	0	0	0	0	0	Y00	Y01	Y02	Y03	Y04	Y05	Y06	Y07	Y08	Y09	M61	M63	M65	M67	M69	M71	M73	M75	M77	M79	0	0	0	0	0	0	0	0	0	0	Y08	Y09	Y10	Y11	Y12	Y13	Y14	Y15	Y16	Y17	M32	M34	M36	M38	M40	M42	M44	M46	M48	M50	0	0	0	0	0	0	0	0	0	0	Y16	Y17	Y18	Y19	Y20	Y21	Y22	Y23	Y24	Y25	LRUN	INTH	+T	-T	S04	S03	S02	S01	Y26	Y27	0	0	0	0	0	0	0	0	0	0	Y24	Y25	Y26	Y27	Y28	Y29	Y30	Y31			<div>Program</div> <div>%99.TXT</div>	
0	0	0	0	0	0	0	0	0	0																																																																																																																
Y00	Y01	Y02	Y03	Y04	Y05	Y06	Y07	Y08	Y09																																																																																																																
M61	M63	M65	M67	M69	M71	M73	M75	M77	M79																																																																																																																
0	0	0	0	0	0	0	0	0	0																																																																																																																
Y08	Y09	Y10	Y11	Y12	Y13	Y14	Y15	Y16	Y17																																																																																																																
M32	M34	M36	M38	M40	M42	M44	M46	M48	M50																																																																																																																
0	0	0	0	0	0	0	0	0	0																																																																																																																
Y16	Y17	Y18	Y19	Y20	Y21	Y22	Y23	Y24	Y25																																																																																																																
LRUN	INTH	+T	-T	S04	S03	S02	S01	Y26	Y27																																																																																																																
0	0	0	0	0	0	0	0	0	0																																																																																																																
Y24	Y25	Y26	Y27	Y28	Y29	Y30	Y31																																																																																																																		
										<div>Instruction code</div> <div>G53</div> <div>T01H0D0</div>																																																																																																															
										<div>Machine Status</div> <table border="1"> <tr> <td>M05</td><td>M09</td><td>M10</td> </tr> <tr> <td>M78</td><td>M33</td><td>M70</td> </tr> </table> <div>G00 X100%</div> <div>F120 X100%</div> <div>S0 X100%</div>		M05	M09	M10	M78	M33	M70																																																																																																								
M05	M09	M10																																																																																																																							
M78	M33	M70																																																																																																																							
										<div>Machine Coord</div> <table border="1"> <tr> <td>X</td><td>0.000 A</td><td>0.000</td> </tr> <tr> <td>Y</td><td>0.000 B</td><td>0.000</td> </tr> <tr> <td>Z</td><td>0.000</td><td>SP360</td> </tr> </table>		X	0.000 A	0.000	Y	0.000 B	0.000	Z	0.000	SP360																																																																																																					
X	0.000 A	0.000																																																																																																																							
Y	0.000 B	0.000																																																																																																																							
Z	0.000	SP360																																																																																																																							
										<div>PartTime</div> <div>0: 0</div>																																																																																																															
										<div>PartNo</div> <div>0</div>																																																																																																															
										<div>SPrpm</div> <div>0</div>																																																																																																															
No Alarm																																																																																																																									
		I/O		ALARM		Reset		CANCEL																																																																																																																	

Alarm information interface

Press “F4(ALARM)”, the first line in this interface shows the number of spindle encoder, the number of current and historical alarm information is record total 10, the superfluous part is clear automatically, only shows 10 alarm information recently.

PLC checking and editing:

Press F3 to check PLC ladder in diagnosis interface, press F5 to edit PLC ladder. Press “S” to search during checking and diagnosis. After modifying PLC, it is to necessary to restart controller, then new PLC will work. You can also press “R” to validate new PLC without restarting.



3.11 Operation of program

Press “Program” in any menu to enter into status of programming.

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

Man Con		N00000		2013-02-26 14:51			
File/File folde in system			Program %99.TXT				
CurDir: /NC			Instruction code				
%99.TXT Z2 LATHER~ MILL TSXT990MDA			G53 T01H0D0				
			Machine Status				
			M05 M09 M10 M78 M33 M70				
			G00 X100% F120 X100% S0 X100%				
			Machine Coord				
			X 0.000 A 0.000 Y 0.000 B 0.000 Z 0.000 SP360				
Compile-P; Receive-R, Tansmit-T, DNC-L			PartTime 0: 0				
			PartNo 0				
			SPrpm 0				
No Alarm							
New/Sek	COPY	RENAM	INFOR	LAST	USBdisk	EXEC	CANCEL

Center part of screen for program display, current program is showed by reverse display, press “PgUp、PgDn” to choose program, and then press “Enter” to edit current program. Functional keys “F1-F7” correspond to: “new file/search”、 “copy”、 “rename”、 “information”、 “last grade” “USB disc”、 “execute program”、 “cancel”.

3.11.1 Editing

Select “New file/search” to popup a dialog box to import the name of program, if the name is existent, the quondam program is called up; If the name is inexistent, the system will build a new file.

The name of program can be number, letter or mix, the length is 100 bits.

The system doesn’t allow the namesake, build a new program or select a program and press “Enter” to enter the editing interface.

Man Con		N00000		2013-02-26 14:51	
N1 M03 N2 G00 X200 Z200 N3 G00 U-44 N4 W-5 N5 G01 U4 W-2 F300 N6 W-11 N7 U-4 W-2 N8 W-5 N9 U10 W-10 N10 W-6 N11 G02 U-6 W-9 I12 K-9 N12 U10 W-15 I25 K0 No.1 /Total27		Program %99.TXT Instruction code G53 T01H0D0 Machine Status M05 M09 M10 M78 M33 M70 G00 X100% F120 X100% S0 X100% Machine Coord X 0.000 A 0.000 Y 0.000 B 0.000 Z 0.000 SP360 PartTime 0:0 PartNo 0 SPrpm 0		No Alarm COMPIL FrELine TeachIn POS DelLine >> CANCEL	

Prompting name of the program in top left corner of the screen; Left is content of the program; Right is the status information of lathe, to carry out operations as follows in editing status:

(1) Locate the current cursor:

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

Press “PgUp” to move the cursor to last page.

Press “PgDn” to move the cursor to next page.

(2) Modify character: Delete the character in the position of cursor, and import new character directly.

(3) Inset character: Import new character directly in the position of cursor. When importing a letter, producing a space automatically in front. When importing a space, importing a letter first and delete the letter.

(4) Delete character: Press “Del” in the position of cursor directly.

(5) Inset a line between the lines: Press “Enter” directly. Inset a line in front of the current line when the cursor in the first line, otherwise inset a line behind the current line.

(6) Editing program:

First page of function:

A, "F1": compile the current program.

B, "F2": to the first or last line of program.

C, "F3": teaching function, enter the handwheel state; this can read the current coordinate of tool according to the corresponding letter X/Z/Y/C/A/B/F4(All axis).

D, "F4": locate to the specified line.

E, "F5": no operation.

F: "F6": delete the current line.

- G, "F7": the first, two page selection.
H, "F8": Chinese character and character input conversion.
The second function:
A, "F1": delete the program.
B, "F2": copy the specified program.
C, "F3": program sort.
D, "F4": search the specified character.
E, "F5": replacing the specified character.
F, "F6": all content to replace the specified character.
G, "F7": first, two page selection.
H, "F8": Chinese characters and character input conversion.

3.11.2 Copy

Press “↑ ↓” in program main interface to select program which need to copy and press “F2” to popup a dialog box to import a new name of program, to copy which is the same content but different name so that to modify, rename and back-up copy.

3.11.3 Delete

Press “↑ ↓” in program main interface to select program which need to delete and press “Del” to delete the program.

The operation of delete need to be careful, it can't be recovery after deleting.

3.11.4 Rename

Press “↑ ↓” in program main interface to select program which need to rename and press “F3” to popup a dialog box to import a new name.

3.11.5 Information

Press “↑ ↓” in program main interface to select program which need to check and press “F4” to popup a dialog box to check the size of program and the remainder space of the system.

3.11.6 Checking program

Press “↑ ↓” in program main interface to select the checking program and press “P”, the system will check the form and grammar of program. Prompting when finding mistake.

3.11.7 Folder management

You can build a file in this system, Press “F1” in program main interface to import a file name and press “.” to build a folder and it will prompt a “folder” after the name.

Move the cursor to the file name and press “Enter” to open to build a new file or folder in it.

Press “F5” go to the last folder.

Move the cursor to the file name and press “Del” to delete the folder.

3.11.8 Select automatical to run program

Press “↑ ↓” in program main interface to select a program and press “F7” to select the program and switch into the last interface.

3.11.9 Program communication

The system could adopt the RS232 serial port to deliver files.

Delivery (Transport)

Deliver the selected program in this system to another system or to PC to save. Press “↑ ↓” in program main interface to select program and press “T” to deliver, press “Reset” to interrupt in the deliver process.

Reception

Receive the selected program in another system or PC (Must be text file form). Press “R” to import a name of received program into the dialog box in program main interface, press “Reset” to interrupt in the receive process.

Pay attention: 1. Using the exclusive communication software to deliver program in User’s PC.

2. The speed of deliver of PC must be the same as the speed of receive, defeat otherwise.

3. The length of RS232 can’t greater than 10 meters.

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

5. Editing program of PC must be text file form.

3.11.10 U-disk management

To exchange files of parameter or program with other system or PC by U-disk. It also can upgrade or back-up the software or parameter in system.

Pay attention: The name of folder can’t have space symbols.

Press “F6” to enter the U-disk management interface in program main interface when U-disk connects the USB port. Press “F6” again to back to the system interface.

Man Con		鑫科数控 N00000		2019-03-19 14:57													
File/File folde in USB disk																	
CurDir: /USB																	
<table border="1"> <tr><td>1000H参数界面 .BMP</td></tr> <tr><td>1000H文件管理 .BMP</td></tr> <tr><td>990H参数界面 .BMP</td></tr> <tr><td>990TPLC界面 .BMP</td></tr> <tr><td>990T参数 .BMP</td></tr> <tr><td>990T参数界面 .BMP</td></tr> <tr><td>990T文件界面 .BMP</td></tr> <tr><td>CURBMP0.BMP</td></tr> <tr><td>CURBMP1.BMP</td></tr> <tr><td>CURBMP10.BMP</td></tr> <tr><td>CURBMP11.BMP</td></tr> <tr><td>CURBMP2.BMP</td></tr> </table>						1000H参数界面 .BMP	1000H文件管理 .BMP	990H参数界面 .BMP	990TPLC界面 .BMP	990T参数 .BMP	990T参数界面 .BMP	990T文件界面 .BMP	CURBMP0.BMP	CURBMP1.BMP	CURBMP10.BMP	CURBMP11.BMP	CURBMP2.BMP
1000H参数界面 .BMP																	
1000H文件管理 .BMP																	
990H参数界面 .BMP																	
990TPLC界面 .BMP																	
990T参数 .BMP																	
990T参数界面 .BMP																	
990T文件界面 .BMP																	
CURBMP0.BMP																	
CURBMP1.BMP																	
CURBMP10.BMP																	
CURBMP11.BMP																	
CURBMP2.BMP																	
Compile-P: Input-R, Output-T, DNC-L																	
No Alarm																	
Program		99.TXT															
Progress		<div></div>															
G53		T0000															
TrueFeed		F 0.000															
M05		M09		M10													
M78		M33		M41													
G00		X100%															
F2000		X150%															
S100		X100% SP000															
Machine Coord																	
X		164.430 A		0.000													
Z		157.845															
C		0.000															
PartTime		0: 0															
PartNo		171															
SPrpm		0															
<table border="1"> <tr> <td>F1 Backup</td> <td>F2 Restore</td> <td>F3 Export</td> <td>F4 Import</td> <td>F5 LAST</td> <td>F6 SYSTM</td> <td>F7 EXEC</td> <td>F8 CANCEL</td> </tr> </table>						F1 Backup	F2 Restore	F3 Export	F4 Import	F5 LAST	F6 SYSTM	F7 EXEC	F8 CANCEL				
F1 Backup	F2 Restore	F3 Export	F4 Import	F5 LAST	F6 SYSTM	F7 EXEC	F8 CANCEL										

A. The processing program management

Copy the files or folder of U-disk into system

After connecting the U-disk, press “F6” to enter the U-disk directory in program main interface. Press “↑ ↓” to move cursor to select file or folder to copy and press “F4” to popup a dialog box to import name, press “Enter” to confirm. If there is the same name of program in the system, it will popup a dialog box to ask if cover the file or folder or not.

Copy the files or folder of system into U-disk

Press “↑ ↓” to move cursor to select file or folder and press “F6”, press “F3” to popup a dialog box to import name in U-disk interface and press “Enter” to confirm. If there is the same name of program in the system, it will popup a dialog box to ask if cover the file or folder or not.

Pay attention: Before unplugging the U-disk must return to the display system of program files directory interface. (Exit U-disk interface)

Otherwise the data which is copied just now will be loss.

The name of folder can't have space symbol when using U-disk.

B. Using U-disk to manage parameter and system software

The system could use U-disk to deliver files or system software to upgrade and update, back-up files and parameter, the method of operation is as follows:

Using U-disk to copy parameter and system software into system(Upgrade, update).

First U-disk inserts the USB port and press “Program” to enter program main interface, press “F6” to show the files in U-disk. Press “↑ ↓” to move the cursor to select a folder which is going to be copied into system and press “Enter” to open it, press “F2” to import code when appearing the files and press “Enter” to confirm, wait for seconds to copied the parameter successfully. Press “F6” to exit U-disk after copying successfully, restart the system, the system will reloads the new files to upgrade the parameter.

Pay attention: The parameter is better to be derived into a separate folder in U-disk to defend from the error operation to destroy the system files.

To derive or back-up parameter files by U-disk

First U-disk inserts the USB port and press “Program” to enter program main interface, press “F6” to show the files in U-disk. Press “F1” to import the code and press “Enter” to confirm, wait for seconds to derive successfully. The parameter in system is already derived into U-disk. Press “F6” to exit U-disk.

Pay attention: The U-disk is empty better to arrange the files (Parameter files is lots of about several dozens) so that derive parameter or create a folder on your computer first, open the folder before deriving to derive the parameters into the folder.

3.11.11 Convert offline file into G code program

Under the program directory select DXF file, press“-” to convert it into G code file, expanded name of G code file is “.CNC”. the input DXF file should use expand name “.DXF” or “.dxf” . it refers to User parameter No.400-No402.

When generate G code file, it will generate the head code and end code according to if there is HEADXF.TEX and ENDDXF.TXT files(headxf.txt and enddxf.txt) in current file directory.

Attention: head code and end code file must be in current file directory.

Chapter 4 Programming

CNC lathe is highly effective automatic equipment according to programmed program to process work piece. Programming is using the CNC system control language according to the requirement and drawing of the work piece to describe the processing trajectory and the assistant action. Ideal system not only could promise process qualified work piece, but also make the functions of lathe reasonable application and fully use, so it is very important to programming, this chapter will introduce many kinds of instructions and usage of CNC program, please read carefully.

4.1 Program coordinate

Absolute programming: it is confirmed coordinates data programming mode based on established absolute coordinate system. X, C/Y, Z, A stand for it.

Example: G00 X200 Z300

When using C to programme, it will execute according to the shortcut if C axis is rotate axis.

Also could use G990 to specify absolute coordinate to programme

Relative programming (incremental programming): is the distance and direction of operation end point, compared with starting point. U, V, W stand for it.

Example: G00 U100 W-200

Also could use G991 to specify the relative coordinate to programme

Mixed programming: is the programming of one segment program by both absolute programming and relative programming.

Example: G00 X200 W30

Diameter programming: all X axis coordinate are presented by diameter.

Minimum program unit: the minimum program unit of XZ is 0.001mm. Real movement of X axis in diameter program is 0.0005mm; real movement of Z axis is 0.001mm.

4.2 Program structure

Name of program: the name of machining program.

Program segment: Be composed of number of line, instruction and data.

Example: N0001 G00 X20 W-300

Mode instruction: the instruction which can remain the function in the program. It works both in this program and program in the future.

In the same operation, there may be several mode instruction, such as M03 (spindle clockwise), M04 (spindle counter clockwise), M05 (spindle stop). They are all Mode used to control spindle. The mode of same kind are categorized into one mode group. At any time it must be one of them, and there is only one of them. The original chosen mode instruction is called mode origin. In the above mode group, M05 is such a mode origin.

Non mode instruction: is the instruction which has no function to store, and only works in the segment of program.

4.3 Program instruction

4.3.1 Functional meaning of address symbol, data list.

Functions	Symbol	Meaning	Data range
File Name		Program file name	0-9,A-Z
Program segment No.	N	Number of program segment	0000-9999
Preparation function	G	Content and mode of designate instruction operation(G01,G02)	00-99
Auxiliary function	M	Auxiliary operation instruction	00-99
Tool function	T	Tool changing instruction	Tool 0000-9999
Spindle function	S	The first spindle speed	0-99999
Spindle function	SS	The second spindle speed	0-99999
Cutting speed	F	Speed per minute, Speed per rotation	1-15000mm/min, 0.001-200mm/r
Coordinates character	XYZA BC	The coordinates value of X C Z A axis	±99999.9999mm
Lead of screw thread	F(I)	F for metric system, I for British system	0.1-1000mm, 1-99teeth/inch
Core coordinates	I K	X Z axes coordinate increment value	±99999.9999mm
Arc radius	R	Arc radius value, tool radius value	0.001-99999.999mm
Delay time	P	G04 delay time	0.001-65s
Program entrance	P	Entrance of calling program segment	0000-99999
Repeat times	L	Times of cycle or subprogram calling L can be used as numbers of multiply screw thread	1-99999 ,1-99
Program skip	/	There is “/” Before N ,this line does not run.	Other parameter P9 could disable this function

4.3.2 Program instruction table

Group	Nature	Code	Functions	Origin	Mode	remarks
1	Main control functions	G00	Rapid point positioning	√	√	
		G01	Linear interpolation		√	
		G02	Circular—clockwise		√	
		G03	Circular—counterclockwise		√	
		G32	spiral interpolation		√	
		G332	G02 spiral			
		G333	G03 spiral			
		G31	Check jump No alarm			
		G311	Check jump alarm			
	Loop command	G70	Finish machining loop		√	
		G71	Cylindrical rough turning loop		√	
		G72	End face rough turning loop		√	
		G73	Enclosed cutting loop		√	
		G74	end face deep hole loop			
		G75	slot loop		√	
		G76	complex screw thread loop		√	
		G90	Circular loop		√	
		G92	screw thread loop		√	
		G93	tap loop		√	
		G94	end face loop		√	
		G22	Program loop		√	
		G800	Cancel loop		√	
	Go to program start point	G26	All go to start point		√	
		G261	X go to start point			
		G262	Y go to start point			
		G263	Z go to start point			
		G264	A go to start point			
		G265	B go to start point			
	Go to G25 point	G61	All go to G25 point		√	
		G611	X go to G25 point			
		G612	Y go to G25 point			
		G613	Z go to G25 point			
		G614	A go to G25 point			
		G615	B go to G25 point			

	Save coordinate	G25	Save current coordinate		√	
	Go home	G28 G281 G282 G283 G284 G285 M800 M881 M882	All axis go home X go home Y go home Z go home A go home B go home C go to spindle encoder zero C axis stop, output M61, check M22 A axis stop, output M63, check M24		√	
	set	G50	Setup coordinate system and max spindle speed of constant line speed		√	
		G52	Setup part coor system		√	
		G184	Setup current Tool coor			
		G185	Setup all Tool coor			
2	Constant line speed	G96	Constant line speed cutting		√	
		G97	cancel	√	√	
	feed	G98	Set feed per minute	√	√	
		G99	Set feed per revolution		√	
	Program mode	G15	Cancel polar coor programme	√	√	
		G16	polar coor programme		√	
		C990	Absolute programme	√	√	
		C991	Incremental programme		√	
		G21	metric system program	√	√	
		G20	British system program		√	
3	delay	G04	Delay time			
4	Running mode	G60	Orientation stop		√	
		G64	Continuum part	√	√	

5	Tool compensation	G40	Cancel tool compensation	√	√	
		G41	Tool in the left of workpiece		√	
		G42	Tool in the right of workpiece		√	
6	Workpiece coordinate	G53	Machine coordinate	√	√	
		G54	Work coordinate 1		√	
		G55	Work coordinate 2		√	
		G56	Work coordinate 3		√	
		G57	Work coordinate 4		√	
		G58	Work coordinate 5		√	
		G59	Work coordinate 6		√	
7	Spindle	M03	Spindle on CW	√	√ √ √	M203 M204 M205 The second spindle rotate CW, CCW, stop in bus type controller
		M04	Spindle on CCW			
		M05	Spindle brake			
		M203	Spindle stop after rotate CW/CCW a while, the			
		M204	time is specified by P, unit:ms			
		M205	Example: M7053 P2000; means spindle rotate CW 2s and then stop			
		M881	C axis stop, output			
		M882	M61, check M22 A axis stop, output M63, check M24			
8	Cooling	M08	Coolant on	√	√	
		M09	Coolant off		√	
9	Chuck	M10	Chuck tighten	√	√	
		M11	Chuck loose		√	

10	Tailstock	M79 M78	Tighten Loose	√	√ √	Control M79
11	Lubrication	M32 M33	Lubrication on Lubrication off	√	√ √	Control M32
12	Huff	M59 M58	Huff on Huff off	√	√ √	Control M59
13	User-defined output	M61 M60	user-define 1 open user-define 1 close		√ √	Control M61
		M63 M62	user-define 2 open user-define 2 close		√ √	Control M63
		M65 M64	user-define 3 open user-define 3 close		√ √	Control M65
		M67 M66	user-define 4 open user-define 4 close		√ √	Control M67
		M69 M68	user-define 5 open user-define 5 close		√ √	Control M69
		M71 M70	user-define 6 open user-define 6 close		√ √	Control M71
		M73 M72	user-define 7 open user-define 7 close		√ √	Control M73
		M75 M74	The spindle servo select the position controlling mode when C axis is output pulse signal, close M75 signal when system starting M03/M04		√ √	Control M75
14	user-defined input	M12 M13	Check M12 valid Check M12 invalid		√ √	Conditional skip, example: M20 P120 means if M12 is valid, program
		M14 M15	Check M14 valid Check M14 invalid		√ √	
		M16 M17	Check M16 valid Check M16 invalid		√ √	
		M18 M19	Check M18 valid Check M18 invalid		√ √	

		M28	Check M28 valid		√	skips to 120 th line. M1810 means check if X10 valid
		M29	Check M28 invalid		√	
		M22	Check M22 valid		√	
		M23	Check M22 invalid		√	
		M18xx	Check Xxx valid		√	
		M28xx	Check Xxx invalid		√	
15	Spindle Gear	M41 M42 M43 M44	first gear second gear third gear fourth gear			To skip when conditions are tenable Example: M12 P120 Program skips to 120th line to carry out.
16	Subprogram	M97 M98 M99	jump call return			L=1-99 P is the line number of transferring program
17	Program	M87 M188 M187 M00 M01 M02 M30 M20	Part number plus 1 Keep feeding and not back home Keep feeding and back home pause Conditional pause, check M22. Program end M05、M09 end Loop start			When other parameter P10=0 is set not to automatical plus 1, instruction M87 to make workpiece number plus 1
18	Spindle	S SS	Speed of the first spindle Speed of the second spindle		√ √	S=0-99999 SS=0-99999

19	Tool	Tab	a means change into a tool b means execute redeem of b		√	a.b=00-12 a=0 means not change the tool b=0 means show lathe coordinate
20	Read the position of absolute motor	M500 M501 M502 M503 M504 M505 M506 M507 M508	M500: read absolute motor position of all the feeding axis and reset the current lathe coordinate. M501: read absolute motor position of X axis and reset the current lathe coordinate. M502: read absolute motor position of Y axis and reset the current lathe coordinate. M503: read absolute motor position of Z axis and reset the current lathe coordinate. M504: read absolute motor position of the fourth axis and reset the current lathe coordinate.			
21	Clear workpiece coordinate	M315 M316 M317 M318 M319 M320	M315:Clear workpiece coordinate of A axis M316:Clear workpiece coordinate of B axis M317:Clear workpiece coordinate of X axis M318:Clear workpiece coordinate of Y axis M319:Clear workpiece coordinate of Z axis M320:Clear workpiece coordinate of all axis			

22	Clear machine coordinate	M415 M416 M417 M418 M419 M420	M415:Clear machine coordinate of A axis M416:Clear machine coordinate of B axis M417:Clear machine coordinate of X axis M418:Clear machine coordinate of Y axis M419:Clear machine coordinate of Z axis M420:Clear machine coordinate of all axis			
23	Instruction of bus driver	M133 M135	M133: Driver rotate according to the specified speed M135: Enable to specify driver			Example: M133 X300; M135 Z0;Close the Z axis driver M135 X1;Enable X axis driver
24	Ratio	M100 M101	Ratio invalid, fixed ratio is 100%; Ratio valid.			

M1xxx: waiting for auxiliary relay effective, for example, M1076 means wait for M76 on;

M2xxx: waiting for auxiliary relay ineffective, for example, M1078 means wait for M76 off;

M3xxx: set auxiliary relay effective, for example, M3330 means set M330 on;

M4xxx: set auxiliary relay effective, for example, M4331 means set M331 off;

M38xx: set output effective, for example, M3809 means set Y09 on;

M48xx: set output effective, for example, M4807 means set Y07 off;

M18xx: waiting for input point Xxx effective then go to next, for example, M1809 means waiting for X09 effective, then go to next step.

M28xx: waiting for input point Xxx ineffective then go to next, for example, M2807 means waiting for X07 ineffective, then go to next step.

M18xx Pxx: check input point Xxx effective then go to next, otherwsie skip to Pxx line. for example, M1809 P234 means if X09 effective, then go to next step, if X09 ineffective, skip to P234 line.

M28xx Pxx: check input point Xxx ineffective then go to next, otherwsie skip to Pxx line. for example, M2807 P456 means if X07 ineffective, then go to next step, if X07 effective, skip to P456 line..

4.4 Programming instruction and usage

4.4.1 Programming stipulation

1. Multiply instruction exist in a segment simultaneously: one program line allows multiply instructions in order to reduce the lines, but the same group of instruction can not share one segment.
2. Within a program segment, instruction and parameters can be arranged optionally. Such as: G01 U10 W-30 can be written: U10 G01 W-30
3. No repeat of instruction within a program segment.
4. It can't be irrelative parameters and operation in a segment.
5. "O" before a instruction is allowed to delete, such as: G01 G03 can be written as G1 G3.
6. The command of optional point, line start or that after tool changing instruction must be programmed by absolute coordinates.
7. Non mode command only in the in specified program line is effective,such as: G61.
8. Mode instruction is always effective before appearing the same instruction.

For example:

N0000 G01 X300 F100; G01 instruction

N0001 X260; G01 instruction

N0002 G00 Z200; G00 instruction, G01 is invalid

4.4.2 Instructions

The basic G mode include: G00, G01, G02, G03, G04, G28, G32, G50

(1) Rapid location(G00)

Tool move to instructive position according to G00 speed in parameter。

As absolute method, use section end point coordinate to program;

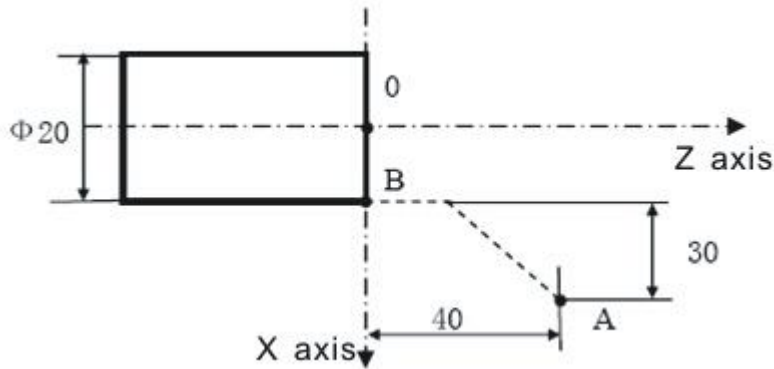
As increase method, use motion distance to program。

Format: G00 X(U)_ Y/C(V)_ Z(W)_ A_

Note: X, Y, Z, A means motion axis。The data point out motion distance and direction by absolute or increase method。

G00 move to aim point according to line way。

Moving speed is determined by parameter。



Example: from A to B.

Absolute program: G00 X20 Z0;

Relative program: G00 U-60 W-40;

Pay attention: The nearest calculation when using absolute coordinate of rotating axis to programme, programme calculation when using relative coordinate;

G00 of every axis is set by parameter, the specified feeding speed with F is invalid. The speed of G00 can be divided into 5%~100%, total six gears, it can be selected by the key on panel.

G00 is mode instruction, when the next instruction is G00 too, it can be omitted. G00 can be written G0.

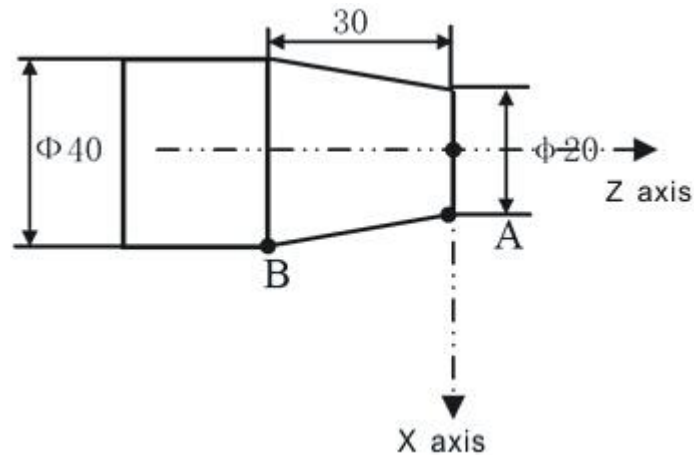
Pay more attention to whether the position of tool is in a safe area or not when X and Z axis both rapid moving, crashing tool in case.

(2) Line interpolation(G01)

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

Format: G01 X(U)_ Z(W)_ Y/C(V)_ A_ F_

Note: X, Y, Z, A means motion axis. The data point out motion distance and direction by absolute or increase method. Motion speed is determined by F word. The F instruction is mode. it will valid till next specifying, so it is unnecessary to specify feed speed in each line.



Linear interpolation

Example: from A to B with F speed.

Absolute program: G01 X40 Z-30 F100;

Relative program: G01 U20 W-30 F100;

G01 also can specify the movement of X axis and Z axis all alone

F feeding speed of G01 could be modified by the feeding rate through the panel, the range is 0%~150%

G01 also can be written G1.

(3) Arc interpolation(G02/G03)

G02 stands for Circular clockwise interpolation, and G03 for Circular counterclockwise interpolation. I is the 2 times of increment of X axis which starting point to center of circle (needless to double it when X axis is radius programming), K is the increment of Z axis which starting point to center of circle, and X Z are the terminal coordinates. It can be also programmed by R not I K.

Format:

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

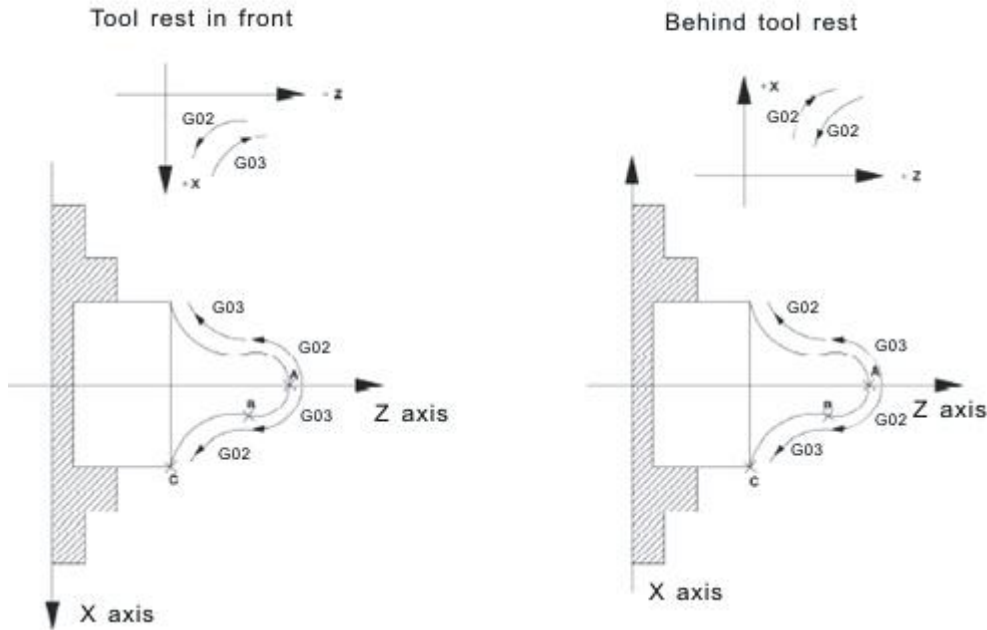
G03 X (U) __ Z (W) __ I__ K__ F__;

G02 X (U) __ Z (W) __ R__ F__;

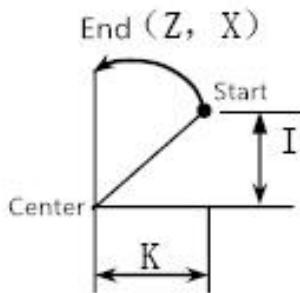
G03 X (U) __ Z (W) __ R__ F__;

Item	Specification		instruction	Meaning
1	Arc direction		G02	Rotate CW
			G03	Rotate CCW
2	End position	absolute	X、Z	Workpiece end position in coordinate system
		relative	U、W	Distance from beginning to end
3	Distance between start point and center		I、K	I(radius)means X, K means Z axis direction
	Arc radius		R	Arc radius (specified radius)
4	Feeding speed		F	Speed along arc tangent of arc

Range of I,K,R: -99999.999~+99999.999



Use X,Z or U,W to specify the endpoint of arc with incremental or absolute value. Incremental value is the distance from start point to endpoint of arc. Center of arc is defined by I,K, corresponding to X,Z. value behind I,K is the vector component from start point(speed parameter=+4 means from arc endpoint) to center, and is incremental value(I is distance, not radius), I and K contains symbol according to position of center.



Arc center also can be specified by radius R, but if arc is over 180°, R should be minus.

Speed of arc interpolation is Fx speed fixing, and is speed of tool along arc tangent direction.

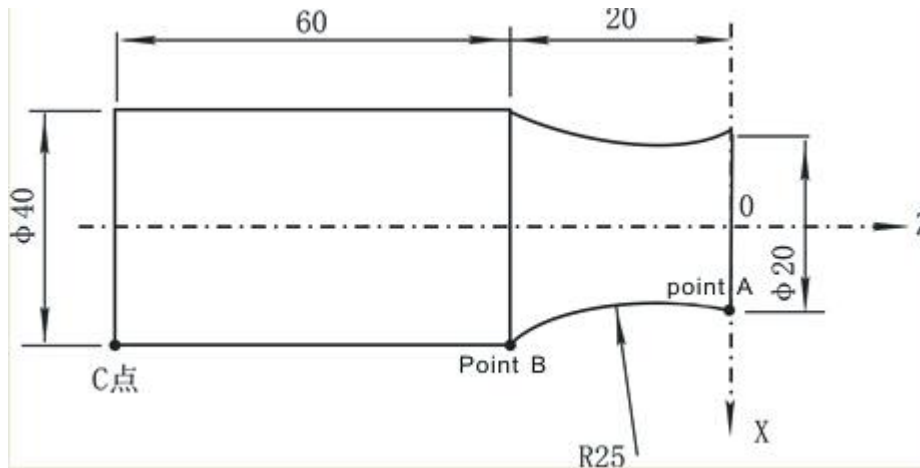
Example: from A to B.

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 .



I0,K0 can be omitted; if I,K,R existed at the same time, R is valid, and I,K are invalid.

Pay attention:

1, Processing arc workpiece usually use ball tool(arc tool) in the actual process, it must use function of tool radius compensation in programming, that's G41 G42 instruction.

2, F is the tool speed along the arc cutting direction.

(4) Screw thread (G32)/Polygonal cutting(G809)

The system can process straight thread, taper thread, continuous straight thread, end surface thread, thread of metric or British system, male and female thread, trapezoidal thread.

Straight thread: just specify Z axis direction and length;

Taper thread: must specify X and Z direction and length;

End surface thread: just specify X axis direction and length;

Continuous thread: several pieces of G32;

Trapezoidal thread: offset angle of spindle in the second loop.

Variable pitch thread: successively input G32, specifying a thread pitch in each segment to vary screw pitch K, it will not detect synchronizing signal of encoder at the beginning of the second loop.

Format: G32 Z(W)- X(U)- F(I)- SP- P-

G32 is mode instruction,

X(U) and Z(W) are absolute or relative coordinate of thread end point. If X(U) is omitted, it is straight thread cutting. If Z(W) is omitted, it is end surface thread cutting. If both X(U) and Z(W) are included, it will be taper thread cutting.

SP(Q): is tool angle at the start point, unit is degree, range: 0-360°; unit of Q is 0.001°. SP defaults as 0.

P: side quantity of polygonal cutting, range:1-12, cutter motor connect with A axis pulse, and set A axis parameter.

F is metric pitch, range is 0.1-1300mm;max pitch=spindle encoder line number/50mm(in the controller with position feedback, namely displacement of long axis when spindle rotate one revolution. max pitch= spindle encoder line

number/10mm)

I is tooth number in British system, range is 1-99 teeth/inch; namely revolution of spindle when long axis move by 1 inch(25.4mm).

During the thread cutting, it will cut on the same route several times from rough cutting to finish cutting. Thread cutting starts from one revolution detection of spindle encoder, so even process cutting for many times, the cutting point on circumference of workpiece is the same one, and route is also the same. Then the speed of spindle is fixed from rough cutting to finishing cutting. If spindle speed varies, it will lead to offset of thread. Besides, at the beginning and end of thread cutting, acceleration and deceleration will cause incorrect section of lead. So instructional thread length should be longer the actual thread length.

Sample1: Straight thread

Thread lead: 4mm

&1=3mm(acceleration section, $&1 \geq 3\text{mm}$).

&2=1.5mm(deceleration section, $&2 \geq 1.5\text{mm}$)

Program according to above data(by twice)

.....

G00 U-62.0;

G32 W-74.5 F4.0;

G00 U62;

W74.5;

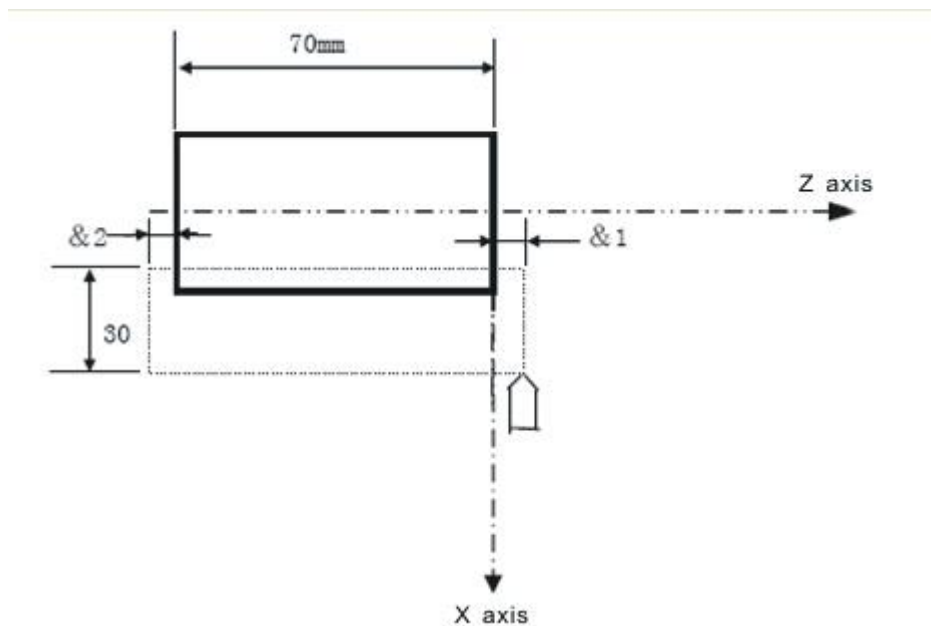
U-64; (the second time cutter further by 1mm)

G32 W-74.5;

G00 U64.0;

W74.5;

.....



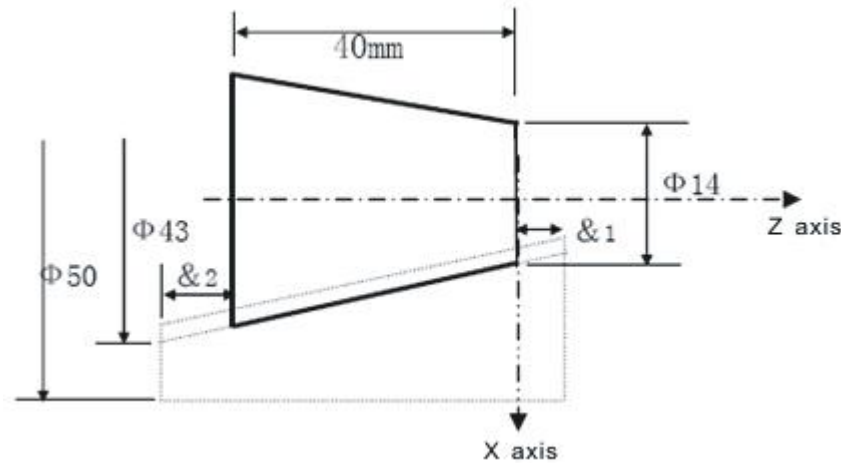
Sample 2: taper thread cutting

Taper thread cutting: ad following picture

Thread lead: 3.5mm

&1=3mm, &2=1.5mm

Program according to above data(by twice)



```
G00 X12 Z3.0;
G32 X41.0 Z-41.5 F3.5;
G00 X50;
Z3;
X10;
G32 X39 Z-41.5;
G00 X50;
Z3;
.....
```

If a program segment is for thread cutting, and next segment is so, then it will not detect one revolution signal at the beginning of cutting, and move immediately.

For example:

G32 W-20 F3; detect one-revolution signal at the beginning of cutting;

G32 W-30 F2; not detect one-revolution signal at the beginning of cutting

Pay attention:

1. When the transmission of spindle and encoder not as 1:1, please modify the Axis parameter P412, P413;

P412 means the number of teeth (requirements: less than or equal to the encoder tooth number, when it is greater than the encoder tooth number, it must match switching board of our company);

P413, the encoder tooth number;

2. In the process of cutting screw, F and rate is invalid.

3. In the process of cutting screw, spindle will not stop whatever you do, if the user want to operate suspend, the system will suspend after processing this segment.

Polygonal cutting loop instruction G809(controller must activate A axis and C

axis);

Format: G809 X(U)_ Z(W)_ I_ J_ F_ P_
X(U)_ Z(W)_

X(U) and Z(W) refer to distance of tool route; I refers to sides quantity; J refers to C axis speed(r/min); F refers to Z axis moving speed(mm/min); P1 means C axis back to zero but A axis does not back to zero; P2 means A axis back to zero but C axis does not back to zero; P3 means both of A and C axis back to zero; if P is omitted, it means neither A nor C axis don not back to zero.

Attention:

1)X and Z is relative to status of G90/G91.

2)If I is positive value, then A axis rotate CW, otherwise A axis rotate CCW.

3)If J is positive value, then C axis rotate CW, otherwise C axis rotate CCW.

4)C axis and A axis needs to enable electronic gear, the higher rotary speed, the bigger electronic gear(numerator is divided by denominator). Or excessive frequency of pulse from controller will lead to step missing in driver.

5)G809 is mode instruction, so in the next sections of fixed polygonal cutting loop, G809, and I,J,F,P can be omitted, till the mode status of G809 is ended by G80/G00/G01/G02/G03/G32.

6)If C axis is required to home before execution of G809, then execute G800. if A axis is required to home, then execute GOTO instruction.

Steps are as following:

- 1)according to P instruction: if C axis did not back to zero, then C axis back home firstly(the same as execution of M800). if back home then rapidly locate at zero point;
- 2)according to P instruction: A axis rapidly locates at zero point.
- 3)long axis of X and Z axis move in the speed of F, while C axis and A axis move in the specified speed proportionally.
- 4)Z axis move to coordinate Z(W), C axis and A axis also stop running.
- 5)Machine coordinate of C axis and A axis execute reduction by 360 degrees.

Polygonal cutting sample:

N1 T0101

N2 G00 X200 Z0

N3 X20

N4 M881 ; C axis back home(M61 connects with spindle orientation, M22 connects with orientation in position)

N5 G150 ; A axis back home(M63 connects with A axis EN, M65 connects with INTH, M28 connects with COIN)

N6 G809 Z-100 I4 J400 F200 ; cutting four faces

N7 Z-105 X30 ; G809 is mode instruction, polygonal chamfering.

N8 G00 X200 Z0 ; cancel G809 mode instruction.

N9 M02

(5) Circularity screw thread(G332、 G333)

Format: G332/G333 Z(W)_ X(U)_ R_ F(I)_ SP_

Use method refer to G02、G03、G32instruction.

(6) Arc in three-dimensional space(G06)

Format:G06 X_ Y_ Z_ I_ J_ K_ F_

Function: If you don't know the center and radius of the arc in three dimensional space. But you already knew the coordination of three point on arc, so you can use G06, the middle point between the starting point and end point to make sure the direction of the arc.

Instruction:G06 is a mode G code;

I:The relative coordination value of the starting point the arc pass through relative to the starting point(X)(radius value, direction);

J:The relative coordination value of the starting point the arc pass through relative to the starting point(Y)(direction);

K:The relative coordination value of the starting point the arc pass through relative to the starting point(Z)(direction).

Pay attention:

- 1).Middle point:Any point except the start point and end point on the arc
- 2).When the three points are collinear, the system alarm.
- 3).I=0 when omitting I,K=0 when omitting K,J=0 when omitting J;the system alarm when omitting I,J,K the same time.
- 4).The meaning of I,J,K is similar to the displacement value I,J,K of the center coordinate relative to the starting point coordinate in G02/G03
- 5).G06 can't process the whole circle.
- 6).This instruction contains large amount of calculation which only can be used with bus system,maybe not smooth with other system;

Example:

```
G0 X10 Y28 Z10
G06 X30 Y98 Z10 I5 J-6 K-5 F100
X130 Y198 Z120 I55 J-86 K-65
G0 X0 Z0
M02
```

(7) Delay Instruction(G04)

Require of work process, delays some time before execute other motion。

Format: G04 P_ or X_ or U_

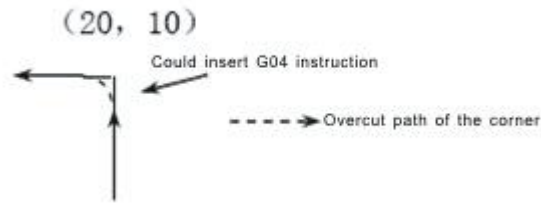
1. P_ unit ms, means delay time。
2. X_ unit S, means delay time, U_ unit S, means delay time。

For example:

```
G04 X1;      delay 1s.
G04 P1000;   delay 1s.
G04 U1;      delay 1s.
```

Special application:G04 can be accurate stop instruction, such as processing corner kinds of workpiece, it appears over cutting sometimes, if use G04 instruction around the corner, it will clear the over cutting.

Example as follows:



Program:

```
N150 G01 X20 Z10 F100;
```

```
N160 G04 P150; (Clear the over cutting)
```

```
N170 G01 W-10;
```

(8) Return Reference(G28)

Return Reference instruction means tool go to reference point according to appointed axis. If there is the mechanical zero point, it will be reference point of machine; if there is no mechanical zero point, the float zero point can be regarded as reference point of machine. It can back to reference point in manual mode, or by G28 instruction.

format: G28 X(U)_ Z(W)_ Y(V)_; ZXYA return to reference across middle 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

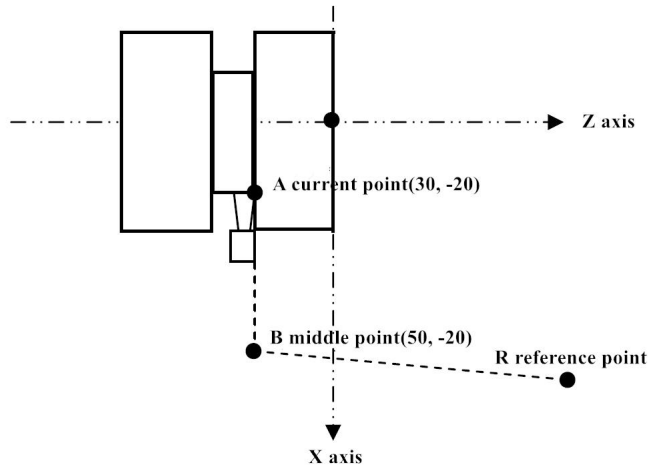
M800 C back to the reference point of encoder, output M75 signal to select controlling position mode when C axis is rotation axis, close M75 signal when system starting M03/M04.

X Z is the middle point of backing to the reference point, use absolute or incremental instruction.

The process:

(1) rapidly locate at middle point from the current point(A B)

(2) rapidly locate at reference point from the middle point.(A B)



Example: return from current point to reference point

G28 X50 Z-20; or

G28 U20 W0;

Attention:

1. if power on, never return reference point manually, then under G28, the time back from middle point to reference point will be the same as the time back to reference point manually.
2. when G28 commands to back reference point, if only one axis's middle point is specified, then only the axis back, another one will not.

(9) Setup work piece coordinate system, max spindle speed of constant linear speed(G50)

Format: G50 X (x) Z (z) ; To set workpiece coordinate

Establish a processing coordinate system according to this instruction, the position of current tool in the processing coordinate system is (X,Z), and this coordinate system is defined as workpiece coordinate system. Once coordinate system is established, the position of following absolute instructional coordinate will be express in this coordinate system. The details refers to operation section.

Pay attention:

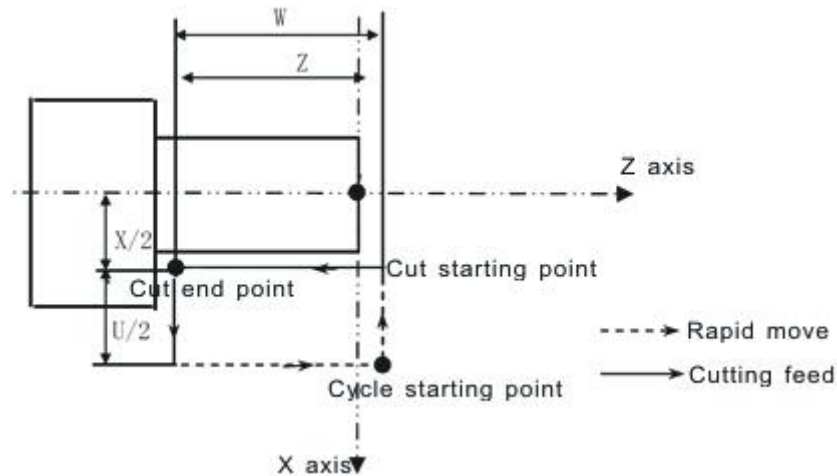
In the status of compensation, if use G50 to set the coordinate, then use G50 to set the position in process coordinate system. Usually cancel the tool compensation first before starting program. Automatically cancel the tool compensation after the system backing to the home point.

G50 S; Set the maximum speed of spindle when cutting constant linear velocity.

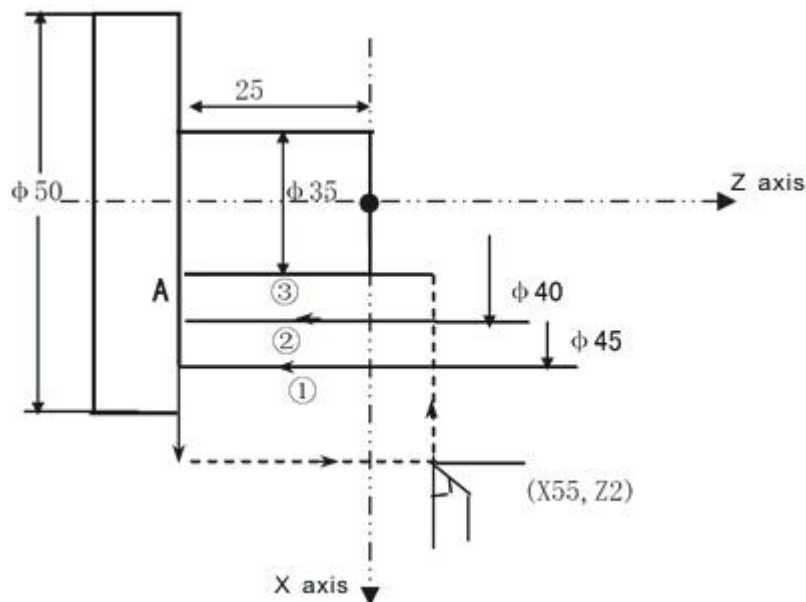
According to this instruction, set the maximum spindle speed of instruction G96 constant line speed in cutting.

(10) Column or taper loop(G90)

Column loop format: G90 X(U)___ Z (W) ___ F___;



Column loop cutting



For example:

```

N10 T0101;
N20 G00 X55 Z4 M03;
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;

```

Every cycle is backing to the starting point at the above program, so cause the situation of cutting endface A again, modify the cycle part program as follow in order to improve efficiency:

```

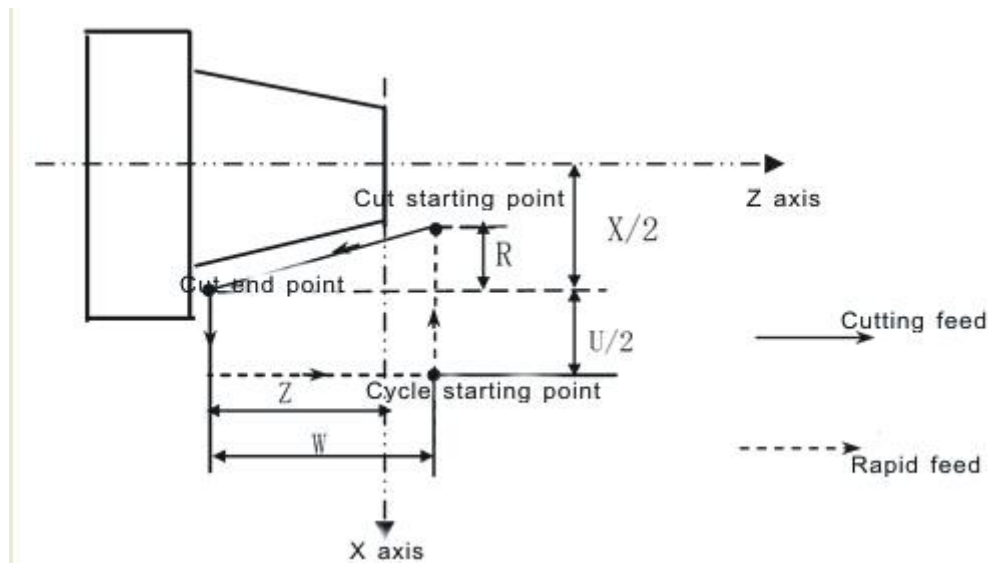
N50 G90 X45 Z-25 F100;
N60 G00 X47;
N70 G90 X40 Z-25;
N80 G00 X42;
N90 G90 X35 Z-25;
N100 G00 ;

```

Taper loop format:

G90 X(U)___ Z (W) ___ R___ F___;

R is radius balance between start point and endpoint of taper. If the coordinate value of start point is less than that of endpoint in the direction of X axis, it will be forward taper with negative R, otherwise it will be backward taper with positive R.



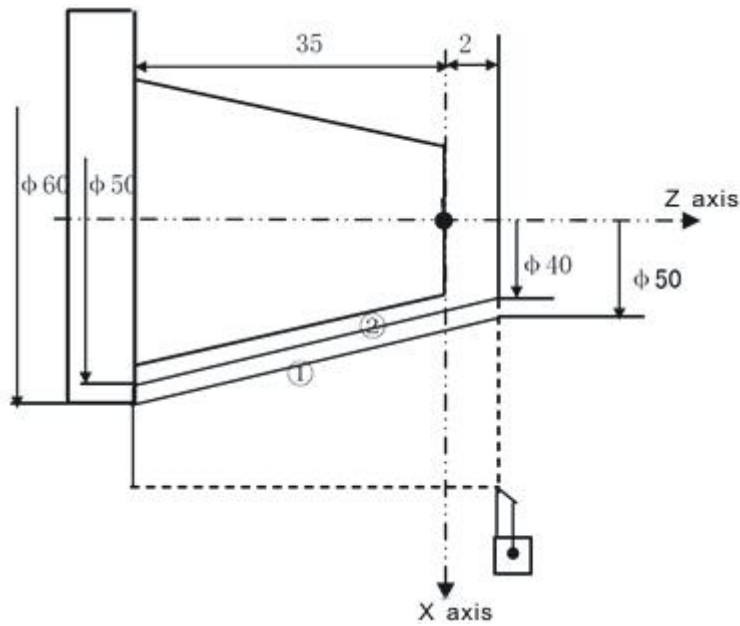
Taper loop cutting

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;

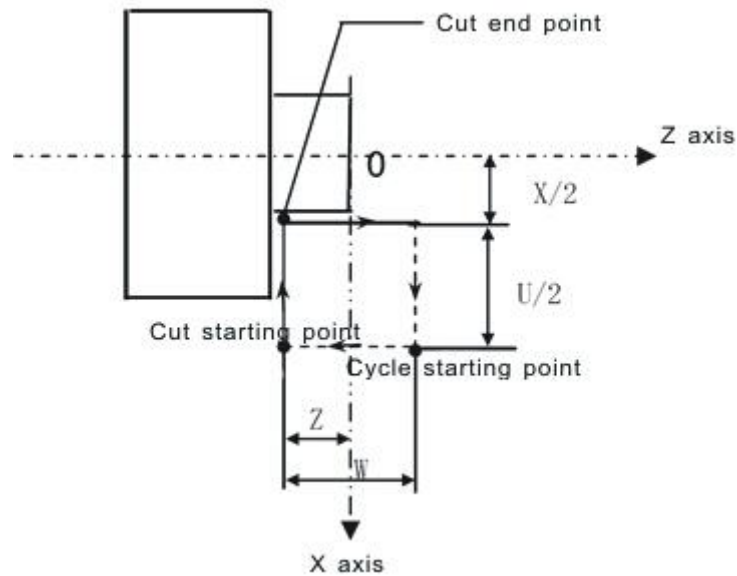
```



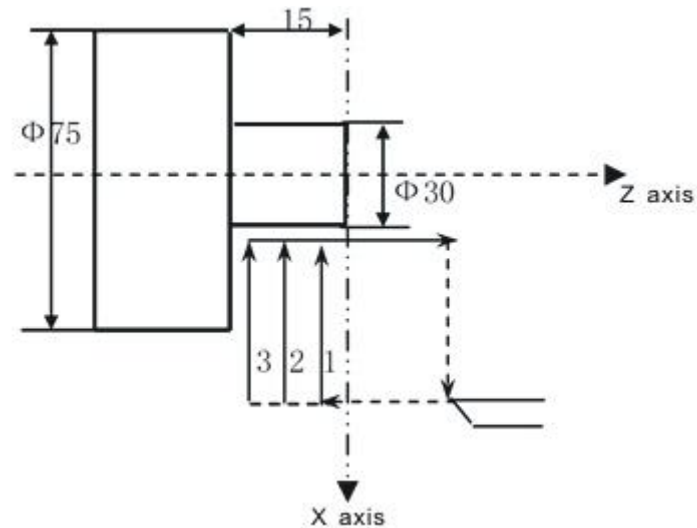
(11) End face loop(G94)

Format: G94 X(U)___ Z (W) ___ F___;

The loop is as following picture, X,Z is coordinate value of cutting end point, U,W is coordinate incremental value of endpoint relative to start pint.



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-10;
N70 Z-15;
N80 G00 X100 Z60 M09;
N90 T0100 M05;
N100 M30;

```

Every cycle needs back to starting point at the above program, make the external diameter parts be cut again, it wastes time, so change the cycle part of program as follow in order to improve efficiency:

```

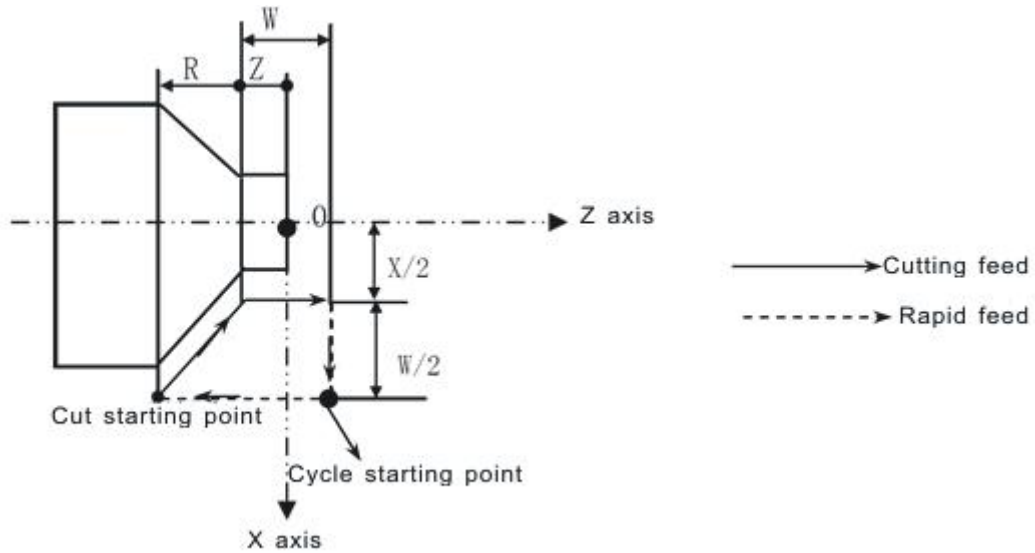
N50 G94 X30 Z-5 F100;
N60 G00 Z-3;
N70 G94 X30 Z-10;
N80 G00 Z-8;
N90 G94 X30 Z-15;
N100 G00 X100 Z60;

```

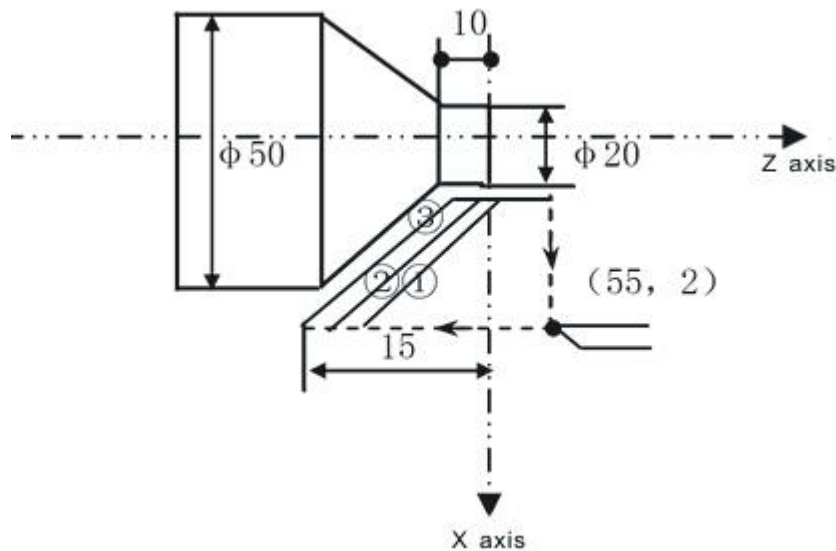
Taper end face loop

format: G94 X(U)___ Z (W) ___ R___ F___;

The loop is as following picture, R is Z axis coordinate component of distance from start point to end point, the track in picture is negative direction of Z axis, R value is minus, otherwise it should be plus.



Taper end face cutting cycle



For example:

.....

N40 G01 X55 Z2 F200;

N50 G94 X20 Z0 R-5 F100;

N60 Z-5;

N70 Z-10;

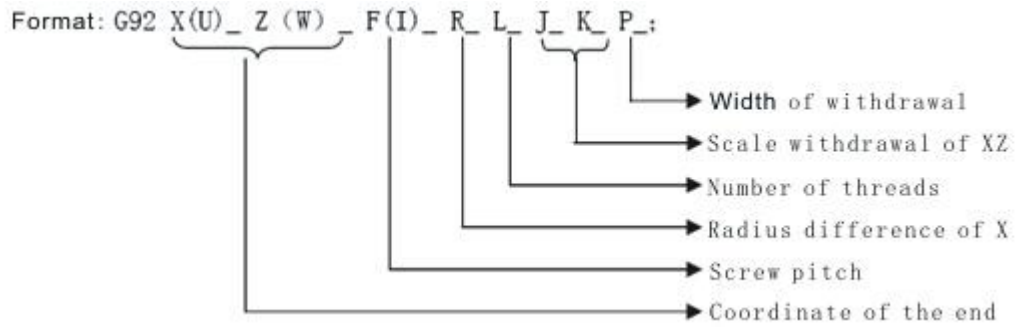
N80 G00 X Z;

.....

R-5 in N50 program: $R-5 = -15 - (-10) = -5\text{mm}$

(12) Screw thread loop (G92)

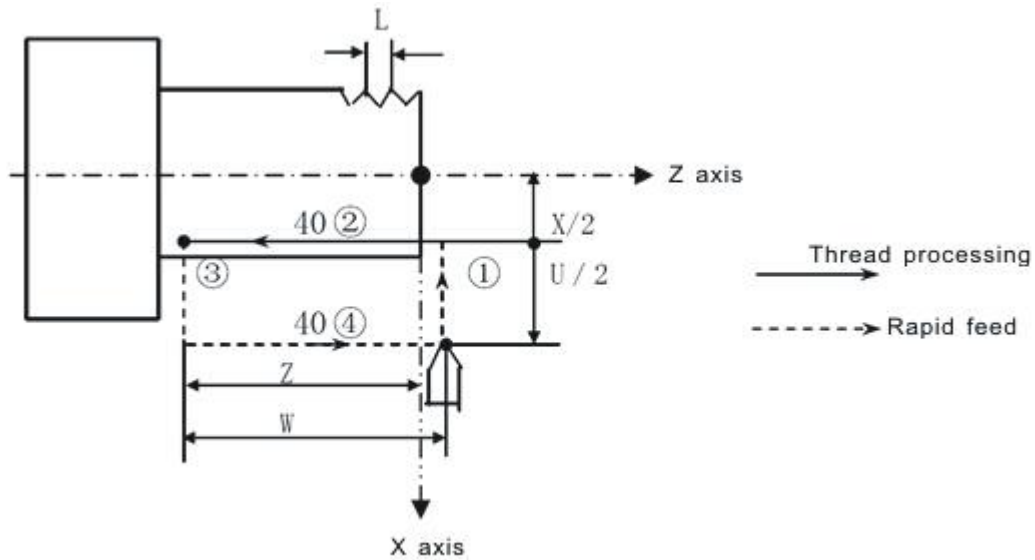
G92 can integrate full loop including from start point "Cut in→thread cutting→cutter relieving→back start point of thread".



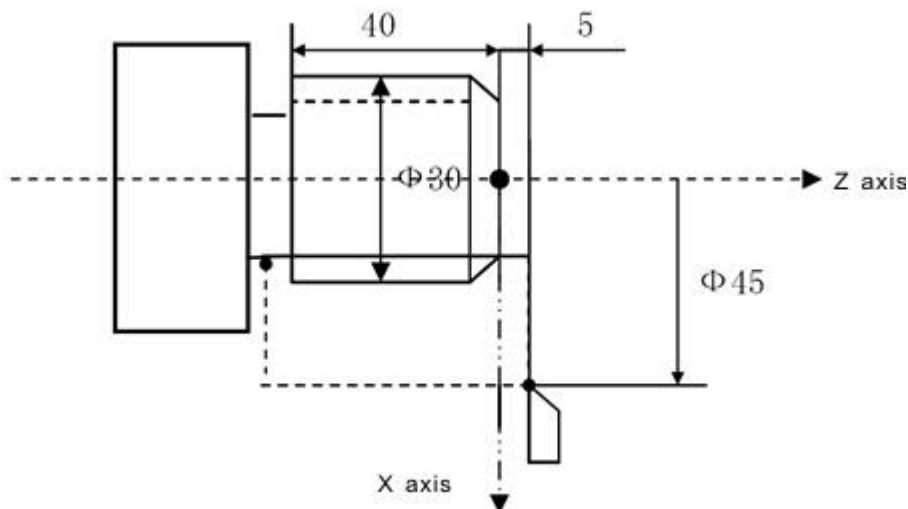
1) Straight screw thread loop

Format: G92 X(U)_ Z (W)_ F/I_;

The process is as following picture. X,Z is the coordinate value of screw end point, U,W is coordinate incremental value of end point relative to start point, dividing into positive and negative. F/I define screw lead L, defined with G32.



Straight screw thread loop



Format:

```

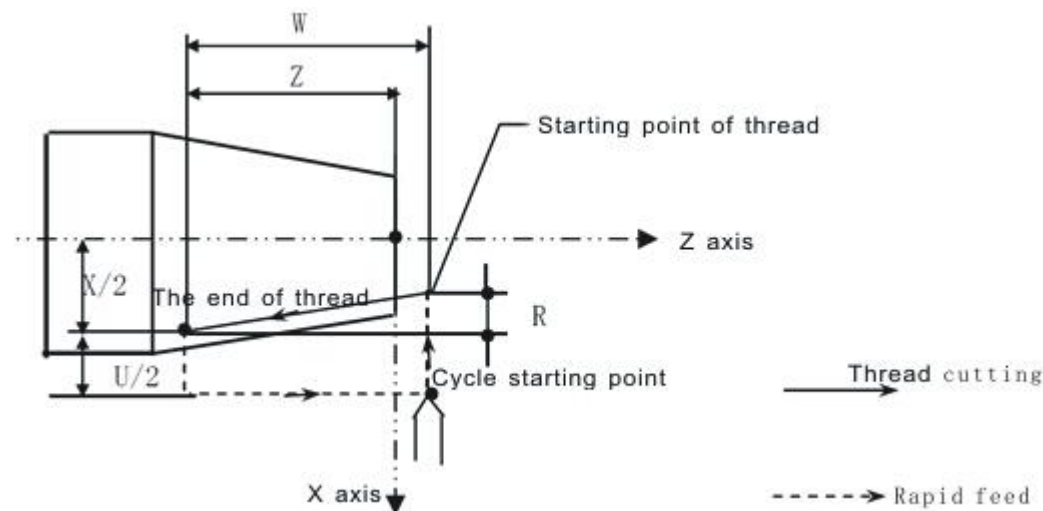
N10 M03 S××;
N20 T0101;
N30 G00 X45 Z5;
N40 G92 X29.2 Z-45 F1.5;
N50 X28.6;
N60 X28.2;
N70 X28.04;
N80 G00 X100 Z50;
N90 T0100 M05;
N100 M30;

```

2) Taper screw thread loop

Format: G92 X(U)___ Z (W) ___ R___ F/I___;

R is radius balance between start point and endpoint of screw cutting in the direction of X axis. If the coordinate value of start point is less than that of endpoint in the direction of X axis, R will be negative, otherwise it will be positive.



Taper screw thread loop

For example: process inner pip tap in british system, pitch 11teeth/inch, taper>1:32.

```

N10 M03 S××;
N20 T0101;
N30 G00 X55 Z10;
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;

Pay attention:

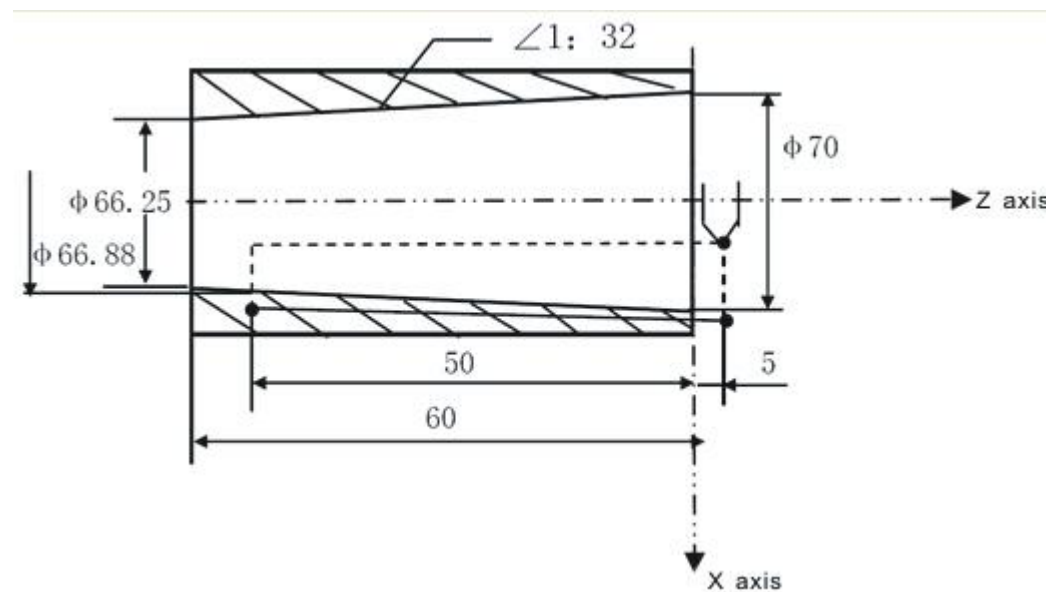
1. When the transmission of spindle and encoder not as 1:1, please modify the Axis parameter P412, P413;

P412 means the number of teeth (requirements: less than or equal to the encoder tooth number, when it is greater than the encoder tooth number, it must match keysets of our company);

P413, the encoder tooth number;

2. When processing inch thread, pitch I is non-mode, just be effective in one sentence, so every sentence should plus I in thread cycle.

3. Spindle speed S is the speed of the processing thread of pitch Fx, system requires this value should be less than 3000mm/min; The back tail speed of X axis is FxSx which is set by speed parameter No.24, system requires this value is less than 5000mm/min, such as: When processing F2, S1200, this value should be less than 20.



3) Deceleration or acceleration control in cutting thread cycle:

At the end of thread, because of the index of deceleration control, cause the distance of pitch is inhomogeneous, the higher speed of spindle the longer of inhomogeneous pitch. To reduce the error, should reduce the index of deceleration or acceleration time, but it will cause the motor stuck if match the step motor. In order to solve this problem:

- could choose Z axis according to linear acceleration or deceleration speed constant;
- could choose the X axis with the rapid speed G00 to back tail.

The relevant parameter is as follows (see the chapter of parameter):

The speed parameter

No.22 the acceleration or deceleration constant of Z axis in thread processing

No.23 the acceleration or deceleration constant of Y axis in thread processing

No.24 The backing tail speed rate of servo motor in thread cycle

No.25 The starting speed of servo motor in thread cycle

No.26 The maximum backing tail speed of servo motor in thread cycle

4) Multi thread processing function, processing function of offset angle

Using L to specify thread number directly. The SP offset angle processing.

Format: G92 X_ Z_ F_ L_ [or SP];

Note:

- Repeat L times of G92 cycle: L thread number. L is mode, after specifying, it will be effective.

- The range of L: 1 ~ 100. Otherwise, according to the L1 to process (single thread).

- The SP (Q) specifies the cutting angle of start, unit of SP is degree, range: 0-360 °, the unit of Q is 0.001 degree. Could not use SP to specify when processing multi thread.

Such as: L03, 3 thread, continuous executing 3 times G92. The first time, begin processing at once when the spindle start rotating, the second time, after 120 degrees offset, begin cutting thread, the third time, after 240 degrees offset, begin cutting thread,

For example:

G92 X50.Z-100 F5 L5; at X50, process 5 thread.

X48.5; at X48.5, processing 5 thread.

X45; at X45, processing 5 thread.

G00 X100 Z100;

.....

5) Thread backs tail when fixed cycle in cutting thread

Program format: G92 X Z F/I P;

P: volume of backing tail: the default value of P could be set by No.20 parameter in User parameter (Default when powering on).

Set unit: P1 means 0.1 pitch ; P10 means 1 pitch .

Scope: 1--225, when the set value beyond to the range is invalid.

6) Back tail at any angle function

When cutting thread without backing fuller, the system must have the function of automatical backing tail in thread processing to produce a qualified section of thread. Including the program format of backing tail in thread:

G92 X_ Z_ F_ J_ K_ P_;

- J, K set the ratio of back tail X, Z. When J2 K1, X is twice faster than Z.

- P: back tail volume. Setting: 0.1 pitch. Set range: 1~255 (beyond to this range is invalid). The default value can be set by No.20 parameter in User parameter (Default when powering on).

- J, K, P are mode value.

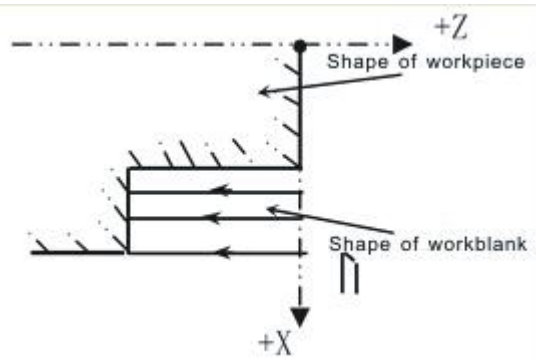
- When executing J0 or K0 in G92, cancel any angle specify, fixed 45 degrees. The default value is 45 degrees when powering on.

- When J K are set to be negative number, or beyond to 65535, it's invalid setting. The range: 1~65535.

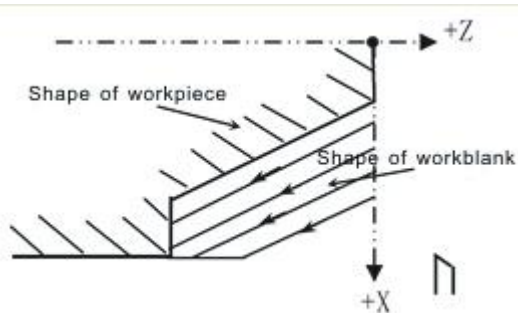
(13) Usage of fixed cycle

Could according to the shape of the workpiece and workblank to choose the suitable fixed cycle. As follows:

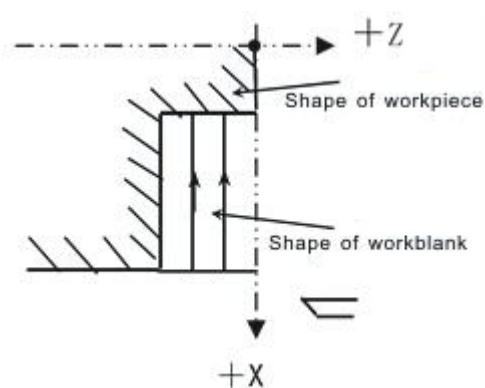
1) Cutting cylindrical cycle G90:



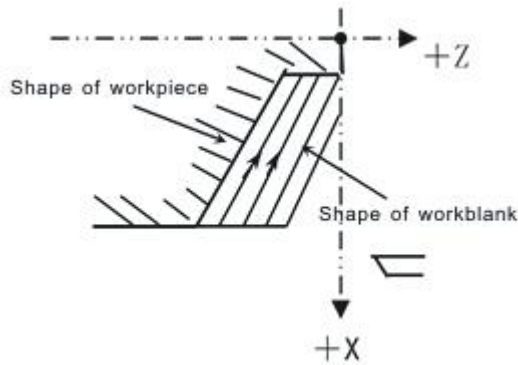
2) Taper cutting cycle G90:



3) Cutting cycle of the end face G94:



4) Taper cutting cycle of the end face G94:



Attention 1: The data X(U)、Z (W) 、R are all mode value in fixed cycle of G90 G92 G94, the front data is always effective when there is not new specified X(U)、Z (W) 、R, except the screw pitch I in inch thread process. Program instruction G90, G92, G94 has been always effective, could use the G code G00, G01 instruction to cancel.

Attention 2: The following three kinds of situation;

①When there is just no moving instruction program behind the fixed cycle,so repeat the fixed cycle.

②It just only use start button to run the program when using MDI.

③In the fixed cycle, if there is M、 S、 T ,then use function of fixed cycle to process together. If cancel the fixed cycle after M、 S、 T(Because of G00 G01), please fixed cycle again.

Example:

```
N10 T0101;
.....
N50 G90 X20 Z10 F100;
N60 G00 T0202;
N70 G90 X20.5 Z10;
.....
```

(14) Fixed cycle of tapping (G93)

G93 Z (W) F/I;

Tapping has two kinds of method: Tracking the spindle encoder (P411=0, spindle must match encoder), spindle servo interpolation of Z axis and spindle (P405=0, P410=92, P411=4). Through the No.404-No.411 parameter in axis parameter. The execution is as follows:

- The first is the same as G32,Z axis feed negative according to the thread cutting.
- Move to the coordinate which is specified by program, stop spindle automatically, after spindle stop completely, the spindle rotates negative automatically according to specification, Z axis returns to the starting position.
- Spindle stop rotating, recover to the direction of spindle rotating in front of the program.

Note:

- The cycle is the same as G90, G93 is a mode code. So G93 should specify G01 or G00 etc..

G93 Z-100 F5; tapping cycle to Z-100;

Z-101; tapping cycle to Z-101;

G00 X50; G00

Tapping method in X axis:

1), No.41 parameter in speed parameter "mode of arc interpolation (0 means A; 8 means B)" is set to 2;

2), Enter G19 into the program segment of X tapping, enter G17 into the program segment of Z tapping, enter G18 into the next program segment after finish tapping.

For example: G93 G19 X-100 F2

G93 G17 Z-100 F2

G18 G0 X30

Pay attention:

- If execute G93 after Z moving positive, the system should execute interpolation negative first because of negative. Should set the parameter of interpolation. If the step motor is stuck, should set the smaller frequency value of interpolate compensation. Or enter the instruction to move the Z axis negative first before executing G93.

- The parameter of the spindle breaking time is affect the start rotating time after stop. Please pay attention to the setting:

Note 1: Z must move negative.

Note 2: Must start spindle rotating before executing G93.

Note 3: Require the spindle breaking time of lathe is short.

Note 4: Require the rotating speed of spindle not to high.

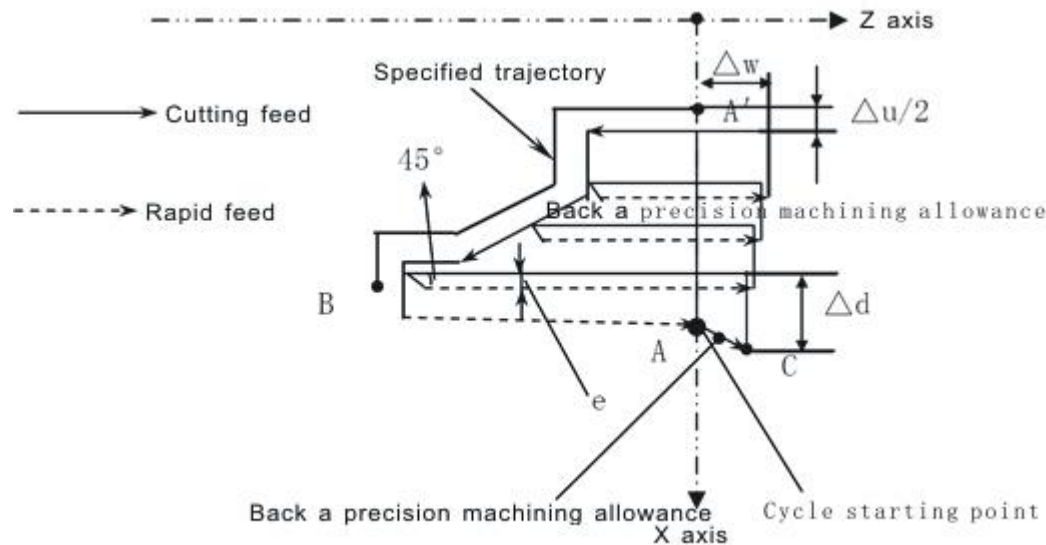
Note 5: For specifying inch thread when specifying I is the same as G32 G92.

Note 6: When choosing the acceleration and deceleration control, if the spindle speed change, there is some delay when making the thread change. So choose the non-speed up or down if require the accuracy. However, with a step motor, the spindle speed can not be too high, otherwise cause the stuck.

(15) Column rough cutting loop (G71)

When use G90, G92, G94, the program is simplified, and there is also some code is defined as compound loop code(G70-G76), simplifying program further. When use these compound loop code, just need to specify the shape of processing, it will perform the full process from rough to finishing.

When command the following shape in the picture, and route $A \rightarrow A' \rightarrow B$, depth Δd , controller will do multicutting parallel with Z axis, finally do finishing cutting according to working allowance Δw and $\Delta u/2$.



Column thick loop

Format:

G71 U (Δd) R(e);

G71 P (ns) Q (nf) U (Δu) W (Δw) F (f) S (s) T (t) ;

Δd : feed thickness, no sign; User parameter P1. mode value;

e: backward distance; mode value, User parameter P2.

i: 1 means start from end face of workpiece. 2 means start from the middle of workpiece, namely the first quarter is a groove. 3 mean no support groove.

J2 means TYPE2 method, J3 means TYPE3 method, otherwise it is TYPE1 method.

TYPE1: start from end face of work piece, namely the first quarter should increase or decrease monotonically;

TYPE2: start from middle part of work piece, namely the first quarter is a groove.

TYPE3: start from end face of work piece, not process groove, X of G71(Z of G72) must increase or decrease monotonically.

ns: first N line.

nf: end N line.

Δu : finishing process allowance in the direction of X axis; User parameter P4. input negative value if inner hole.

Δw : finishing process allowance in the direction of Z axis; User parameter P5.

Note 1: G71 in the use of rough machining cycle, only F S T in the program G71 function is effectively, but F S T in program of ns→nf is effectively to fine machining, invalid in rough machining cycle.

Note 2: A—B must conform the increase or decrease together mode of X and Z axis.

Note 3: G97 G96 is invalid for rough machining cycle when the program

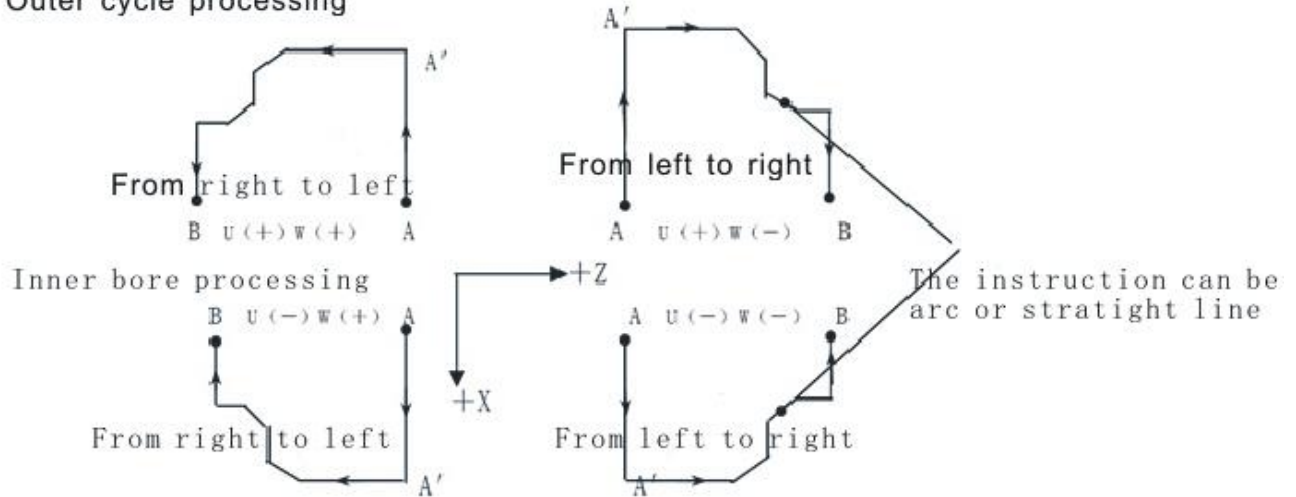
ns→nf includes choose function of constant linear velocity, G96 G97 in the G71 or front program segment is effective for rough machining cycle.

Note 4: from A to A', maybe G00 G01 is in the NS program segment, but not containing Z axis movement instruction.

Note 5: In program segment of NS to NF, can't call subprogram.

Note 6: Use G71 to cut shape, four method as follows, the four mode is cutting according to the tool which is parallel Z axis, Δu , Δw finishing machining allowance, as shown below:

Outer cycle processing



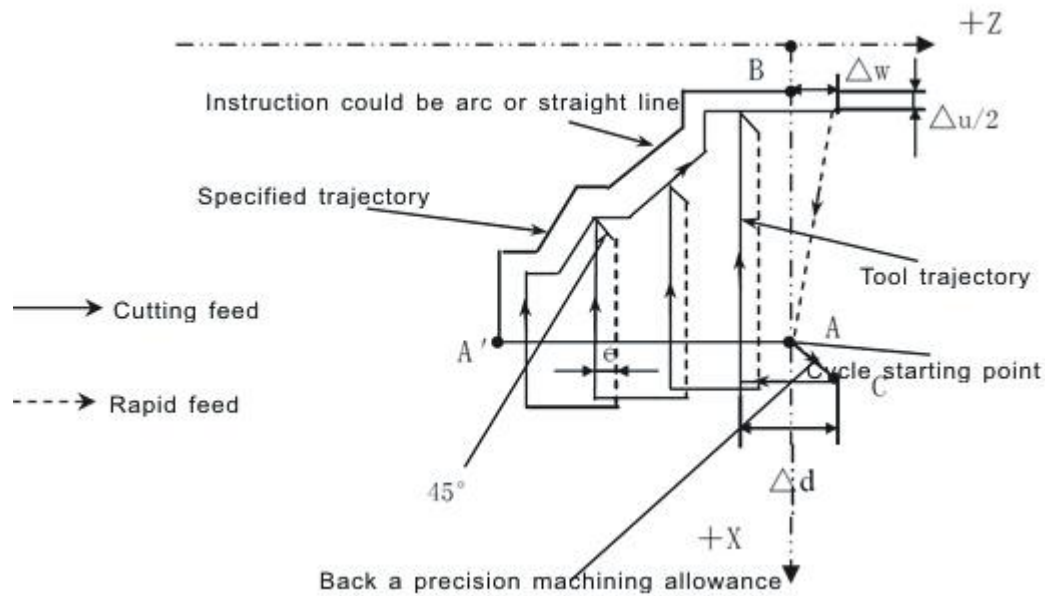
(16) End face thick loop(G72)

G72 and G71 are rough machining cycle instruction, but G72 is along the direction which is parallel X axis to process cutting cycle, format:

G72 W (Δd) R (e);

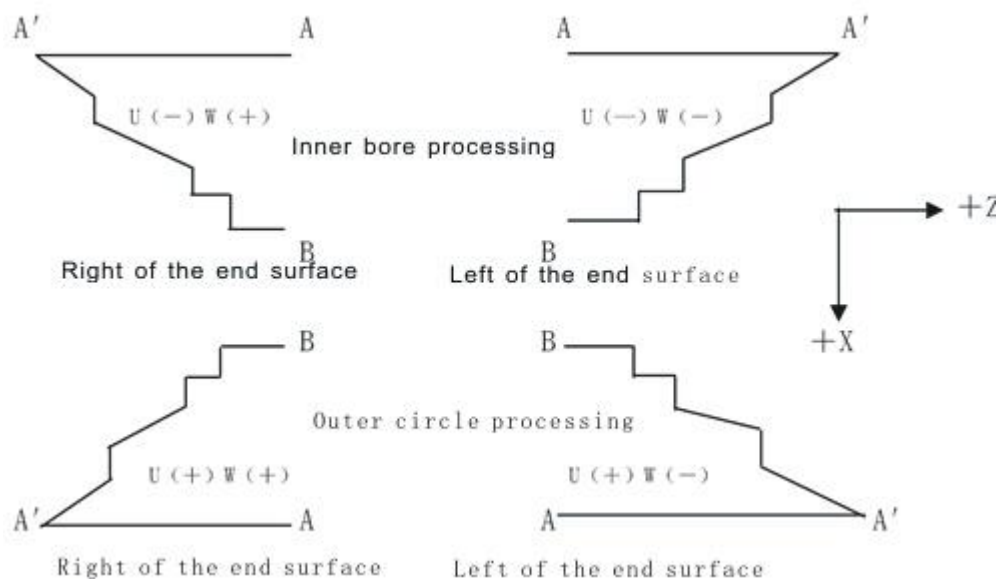
G72 P (ns) Q (nf) U (Δu) W (Δw) F (f) S (s) T (t) ;

The parameters Δd , e, ns, nf, Δu , Δw , f, s, t and G71 have the same meaning



Rough machining cycle of end surface

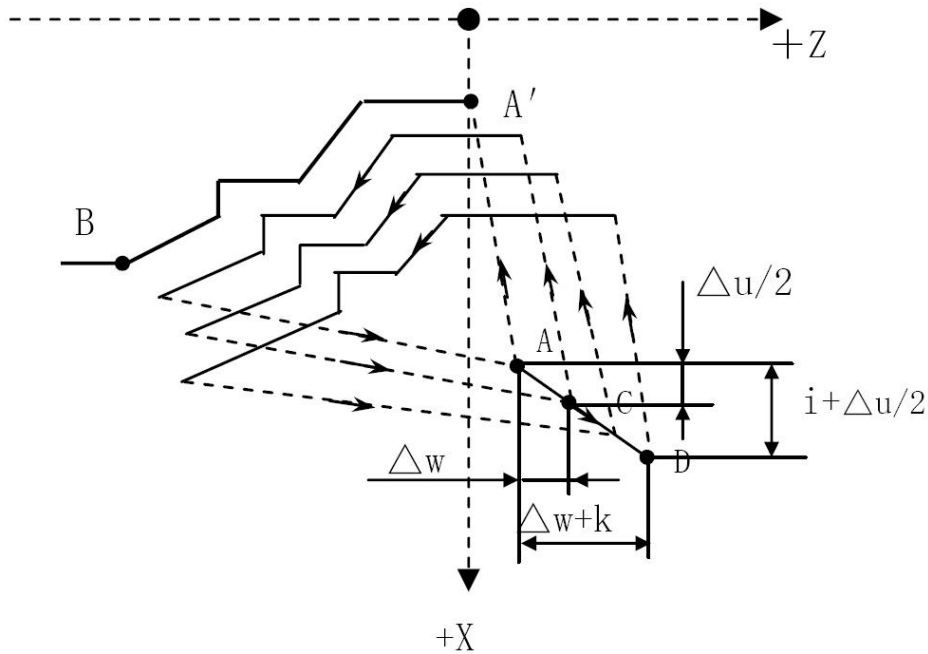
Use G72 to cut the shape, there are four situation. No matter what kind of is the tool parallel the X axis to cut again. Δu , Δw symbols are as follow



Between A and A', the program segment ns could contain G00 or G01, but can not contain X axis instruction, between A' to B, the X and Z axis must be all increase or decrease graph, that's a direction to increase or decrease.

(17) Close cutting cycle(G73)

The closed cutting cycle according to the cutting shape to be close to final shape. This way is high effective to cast or cut workblank. G73 cycle is shown as follow:



Format:

G73 U (i) __ W (k) __ R (d) __;

G73 P (ns) __ Q (nf) __ U (Δu) __ W (Δw) __ F (f) __ S (s) __ T (t) __;

N (ns); --\

.....; |

. > route of A→A'→B finishing machining, segment from ns to nf to instruct.

. |

. |

. |

N (nf); --/

i: distance and direction of backing tool in the direction of X axis; mode instruction; User parameter P7.

k: distance and direction of backing tool in the direction of Z axis; mode instruction; User parameter P8.

d: cutting times; mode instruction; User parameter P6. if define R(d)=1 means rough processing time is 1.

Other parameters are same as G71. difference of G73 from G71, G72 are F,S,T only effective in rough machining.

Note 1: The cycle is according to the program which is between P and Q in G73. The tool backs to A point automatically after finish cycle.

Note 2: Increase or decrease X or Z axis is invalid when using G73.

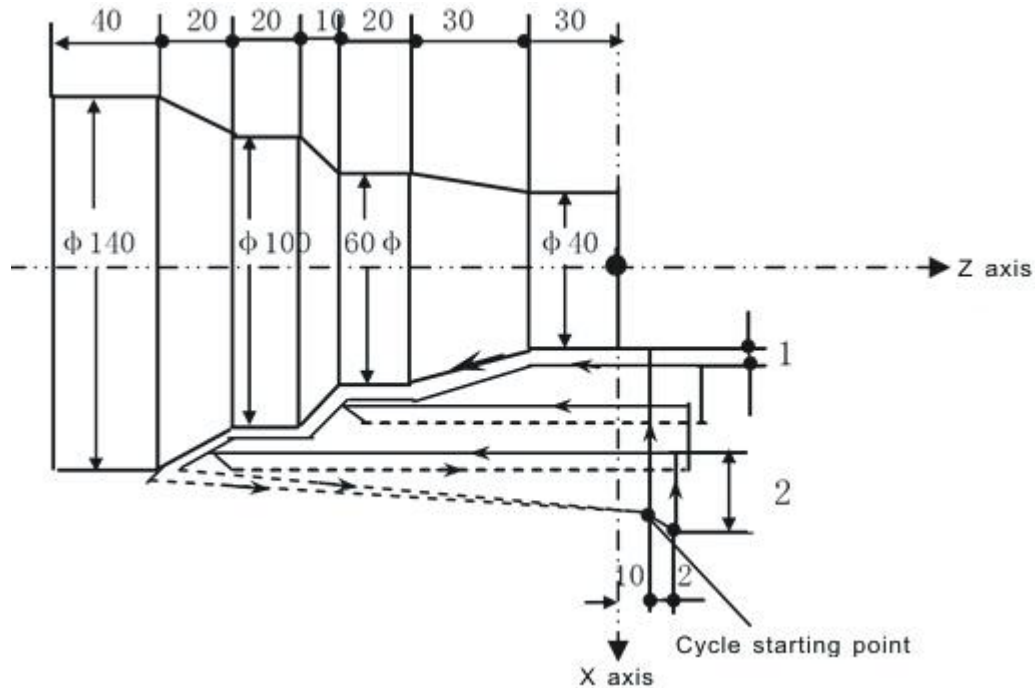
(18) Finishing machining cycle (G70)

Could use G70 to fine machining after using G71 G72 G73 to finish rough machining.

Program format:

G70 P (ns) Q (nf) ns and nf are the same as the above.

F、S、T are all invalid in G71 G72 G73 when fine machining, F、S、T are effective only in ns→nf. Tool use rapid feeding to back to starting point when G70 is over and start read the next program segment of G70 cycle.



N10 M03 S1500;

N20 T0101;

N30 G00 X160 Z10;

N40 G71 U2 R1; in rough machining loop, cut 2mm and back 1mm each time single side in the direction of X axis.

N50 G71 P60 Q120 U2 W1 F100 S2000; the final route of rough machining is from N60-N120. and leave finishing machining allowance X axis 2mm, Z axis 1mm.

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;

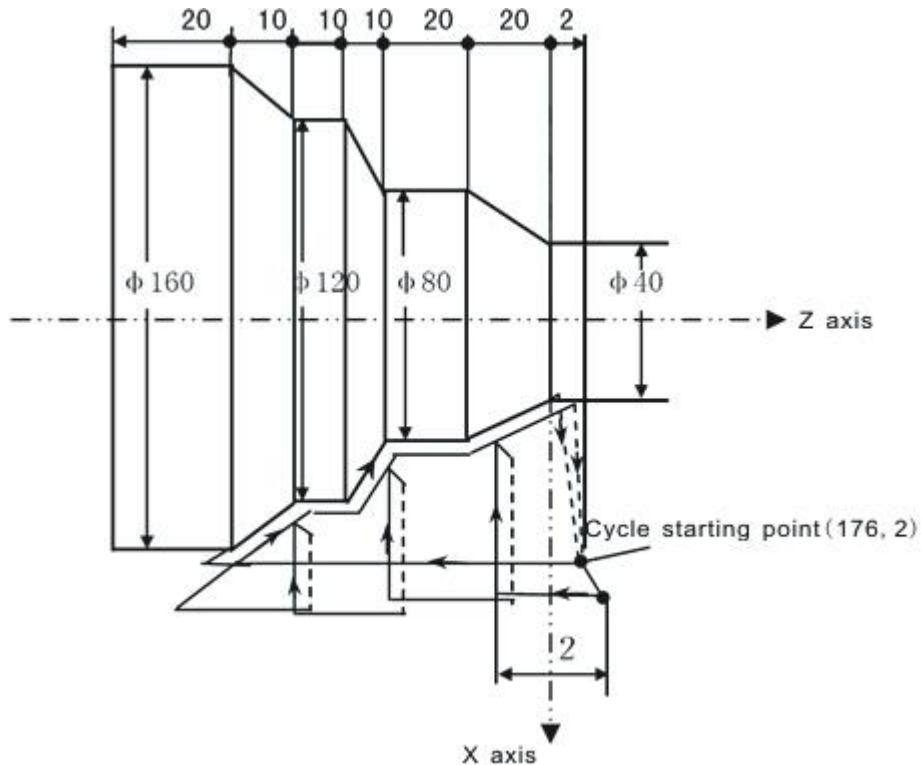
N130 G70 P60 Q120; specify route of finishing cutting.

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; in rough machining loop, cut 2mm and back 1mm each time

N50 G72 P60 Q120 U2 W1 F100 ; the final route of rough machining is from

N60-N120. and leave finishing machining allowance X axis 2mm, Z axis 1mm.

N60 G00 Z-72;

N70 G01 X160 Z-70 F80;

N80 X120 W10;

N90 W10;

N100 X80 W10;

N110 W20;

N120 X36 W22.08;

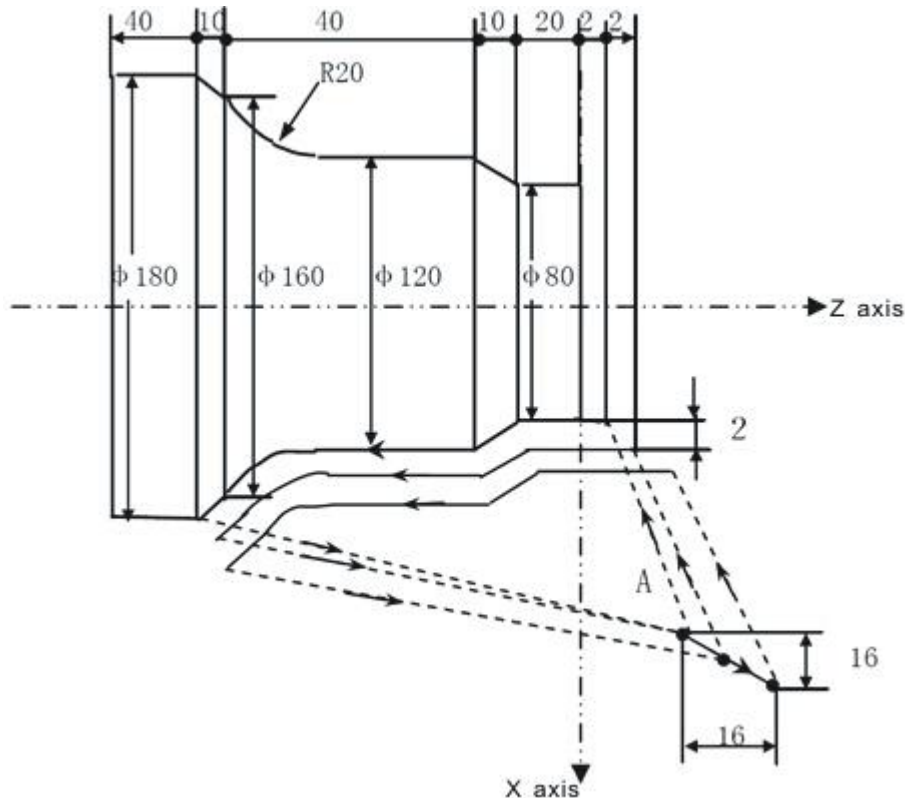
N130 G70 P60 Q120;

N140 G00 X200 Z50; specify route of finishing cutting.

N150 T0200 M05;

N160 M30;

For example3: G73 G70



```

N10 M03 S3000;
N20 T0303;
N30 G00 X220 Z40;
N40 G73 U14 W14 R0.010; rough cutting allowance: X axis 14mm, Z axis 14mm, 10
      times.
N50 G73 P60 Q110 U 4 W2 F100; specify rough cutting route according to
      N60-N110, leaving finishing cutting allowance:
      X axis 4mm(diameter), Z axis 2mm;

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; specify route of finishing cutting.
N130 G00 X250 Z50;
N140 T0300 M05;
N150 M30;

```

(19) End face deep hole loop(G74)

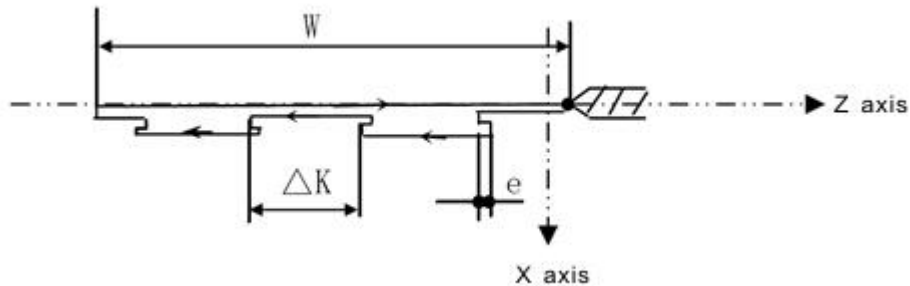
Format:

G74 R (e) ;

G74 X (u) P (Δi) Z (w) Q (Δk) F (f) ;

e: backward distance; mode instruction, User parameter P10. unit:mm.

Z (w) Z depth; unit:mm
 X (u) X end-point coordinate;
 Δk : Z feed thickness;User parameter P9.
 Δi : X feed thickness.
 F: feeding speed



For example:

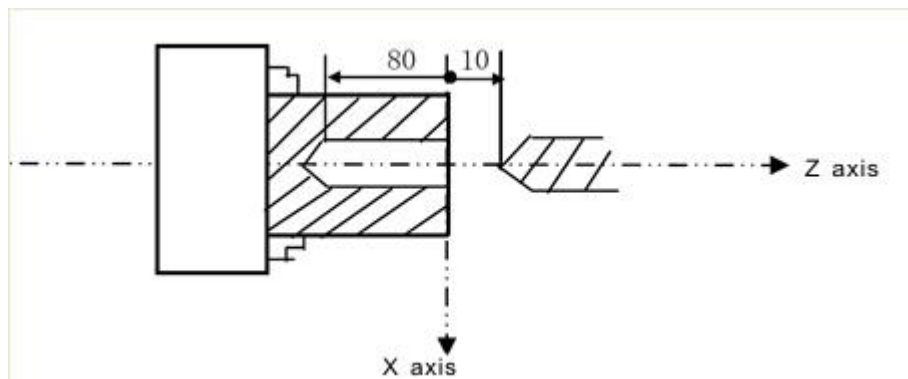
```
N10 G00 X0 Z10;  

N20 G74 R2;  

N30 G74 Z-80 Q10000 F800;  

N40 G00 X50 Z50;  

N50 M30;
```



(20) Slot cutting loop(G75)

Format:

G75 R(e)___;

G75 X(U)___ P(Δi)___ Z (w) ___ Q (Δk) ___F(f)___;

e: backward distance;User parameter P10.

X (u) groove depth;

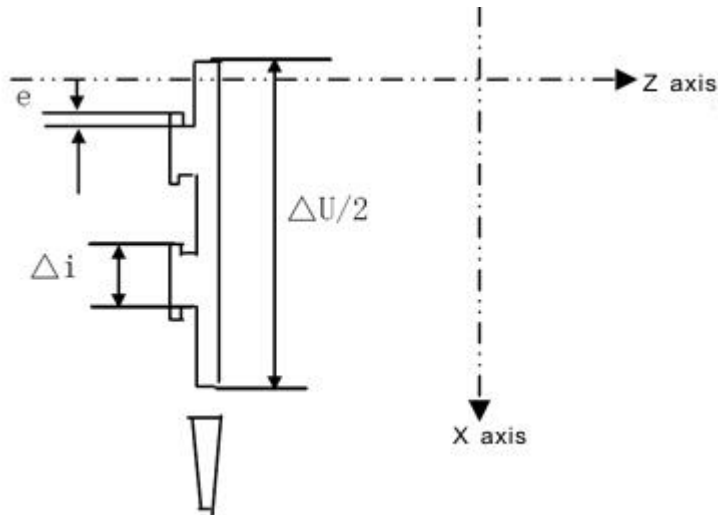
Z (w) Z end-point coordinate;

Δi : X feed thickness;User parameter P9. radius value.

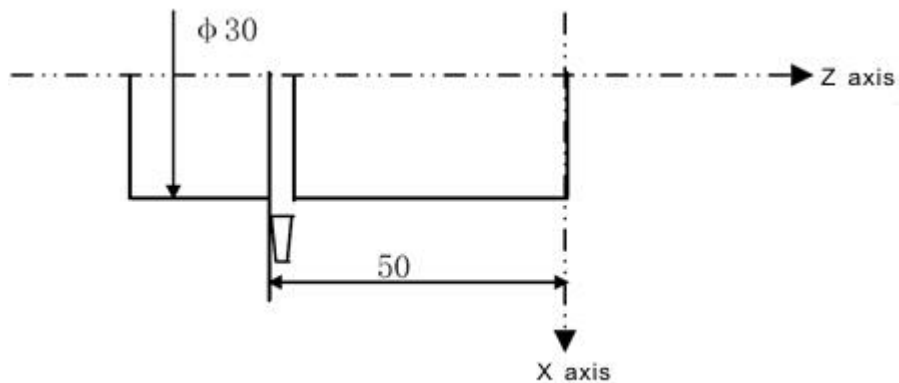
Δk : Z feed thickness.

F: feeding value.

Circle is like following picture:



Sample:



For example:

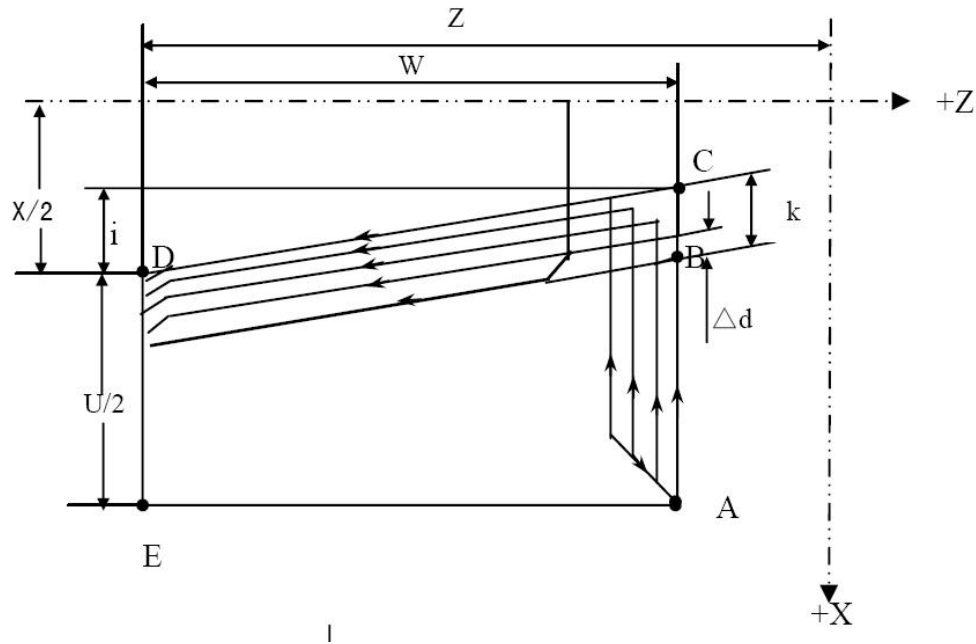
```

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;

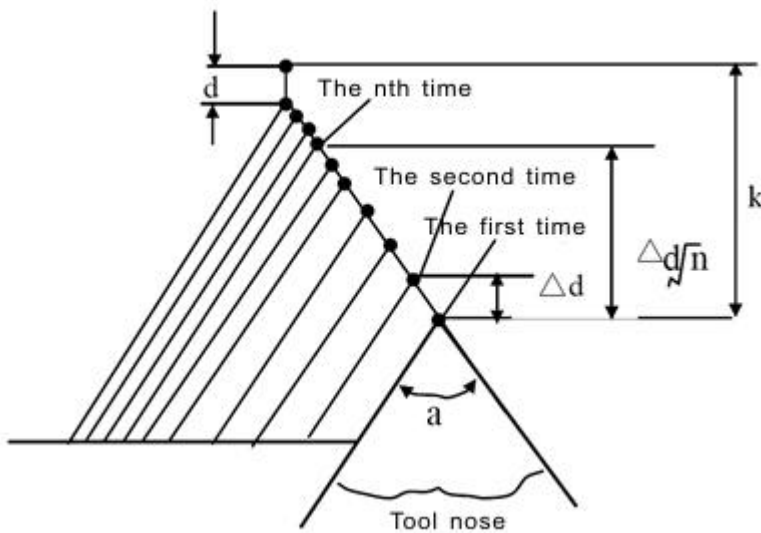
```

(21) Complex screw thread cycle(G76)

According to following program, the screw cutting is as picture:



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——equidistance feed;
 2——If the fist feed is too long in degression feed,so divide into two feed.
- c: 0——right enter;
 1——left enter;
 2——middle enter
 3——right and left enter, the first feed is middle.
- m: finish turn times, mode instruction, User parameter P11.
- r: chamfer length,User parameter P12.

a: thread tooth angle(degree),User parameter P13.

b, c, m, r and a can be defined by P at the same time.

For example, when b=2, c=3, m=1, r=1, a=60 defined as follows(K is pitch):

P b/2 c/3 m/01 r/10 a/60 can be programmed as P23011060.

Δd min: minimal cutting depth,User parameter P14. when one-time cutting

depth($\Delta D \times \sqrt{N} - \Delta D \times \sqrt{N-1}$) is smaller than Δd min, Δd min will be one-time cutting depth. it is mode instruction. Unit is um. Radius value.

d: finish turn allowance,User parameter P15. it is mode instruction. Unit is micron.

Radius value

I: X taper screw thread feed measure.

k: height of thread(X distance is radius instruction). unit is um.

f: metric lead. Unit is mm.

L: multiple thread head numbers. When define L, SP is ineffective.

SP(Q): start angle: 0-360°; Q unit is 0.001 degree. If no defined SP, default as 0.

Δd : the first time cutting value(same sa G32), unit: um, or tool feed times. The function of Q(Δd) is defined by User parameter P24. when P24=8, address Q(Δd) means rough processing times, default value is 1; otherwise address Q(Δd) means the first time cut depth, when Q(Δd) define feeding times, it has equidistant mode and decreasing mode.

When tool width is smaller than groove width, offset a fixed position gradually to excute G76 to finsh process.

#201: repeat times of G76, when #210 is positive, it means after run #201 times, then cut alone X direction, in order to remove chips easily. When #201 is negative value, it means after finish G76 one time, then offset another position to repeat G76, but chips removal may not be well.

#202: tool offset value in Z direction when G76 repeats. Uint:mm

#202 can be positive or negative, means different offset direction. Actually, controller will calculate offset angle of tool start point accoring to Z axis offset distance. Then each time when run program, controller will clear #201 and #202. so when proramme, add two assignment statement into program, for example:

G0 X_ Z_;

#201=3; repeat G76 three times.

#202=2.5; offset 2.5mm in Z direction when repeat G76.

G76 X_ Z_

G0 X_ Z_;

.....

Feeding value and rough turning feeding times in all cases are as follows:

1) b=0, P24≠8, each time feeding depth is $\Delta d \sqrt{n}$;

2) b=0, P24=8, each time feeding depth: method is the same as 1), $\frac{(k-d)}{\sqrt{\Delta d}} \sqrt{n}$;

3) b=1, P24≠8, feeding value is Δd , rough turnning feed times is $(k-d)/\Delta d$;

4) b=1, P24=8, feeding value is $(k-d)/\Delta d$, rough turnning feed times is Δd ;

Attention:

Circle movement will run through G76 defined by address X(U), Z(W), unit:mm.

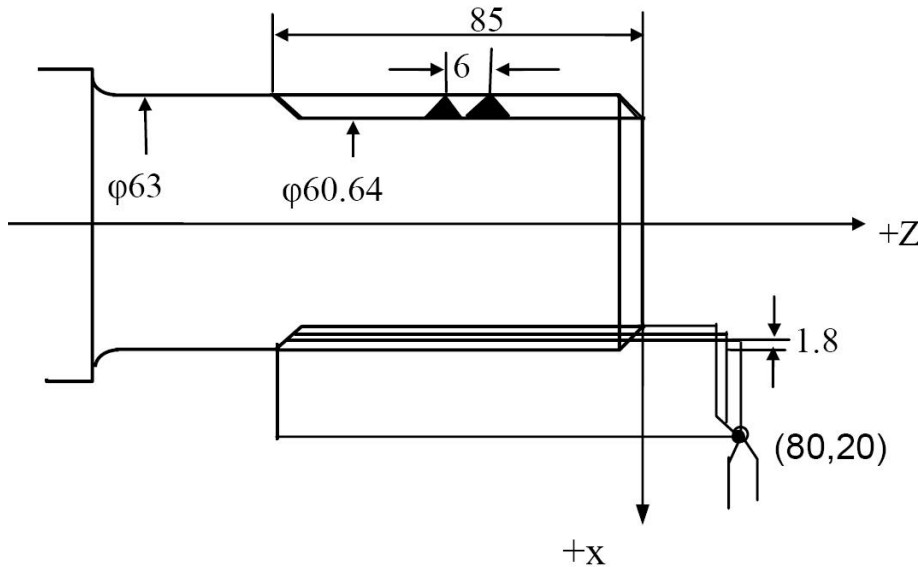
In above circle of picture, incremental symbol is as follows:

U,W: negative(defined by direction of track from A to C, from C to D);

R(I): negative(defined by direction of track from A to C);

P(K): positive(always);

Q(Δ d): positive(always);

Compound fixed loop G76 sample:

```

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;

```

Note 1: pay attention to cut thread, use G32 to cut is the same as using G92.

Note 2: specify the chamfering amount of thread, it's also effective to the G92 thread cutting circle.

Note 3: when matching step motor, because of the acceleration or deceleration the thread in tail will be inhomogeneous. So should choose the linear acceleration or deceleration to control X axis with G00 to back tail fast.

Pay attention to compound type of fixed cycle (G70 ~ G76):

(a) In the specified compound type of fixed cycle program segment, P, Q, X, Z, U, W, R and others necessary parameter must be specify directly in each program segment.

(b) In the program segment of G71, G72, G73 , if there is a P specify the sequence number, then the program segment corresponding to the sequence number must be the G code of G00 or G01.

(c) In the MDI mode, can not execute G70, G71, G72, G73 instruction. G74, G75, G76 can be executed.

(d) in the program segment of G70, G71, G72, G73 and the program segment between P and Q, could not specify M98/M99.

(e) In the program segment of G70, G71, G72, G73, can not have the following instruction within the program segment which is specified by P and Q.

★ except the G04 (pause) one-time code

★ except the G00, G01, G02, G03 code

★ M98/M99

(f) in the implementation of compound fixed cycle (G70 ~ G76, can make the action to stop inserting manual movement, but to start the execution of compound fixed cycle again, must return to insert a manual before motion position. If you do not return again, the mobile manual is not in the absolute value, the action of dislocation, and its value is equal to the quantity of mobile manual.

(g) G70, G71, G72, G73, P, sequence number specified in Q, not be coincident in this program.

(h) In G70, G71, G72, G73, use P Q to specify the program segment of fine machining shape, the last motion instruction could not be chamfer or transition round.

(22) Program loop(G22、 G800)

G22 is a cycle instruction of program, G800 is the end of the cycle instruction. The two must be used together, be used for the situation of repeat process. L is the cycle times, the range is 1-99999. The cycle instruction can be nested.

Format: G22 L-

Loop segment;

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-11 U6	
N0006	W-30	
N0007	W-10 U5	
N0008	G0 U10	
N0009	W51	
N0010	G800	; loop end
N0011	G26	
N0012	M30	

(23) Part coordinate setup(G52), Setup or offset tool coordinate (G184、 G185)

When programming in the workpiece coordinate system, to set sub coordinate

system in order to programme.

Format:

G52 X_Z_; absolute coordinate setup

G52 U_W_; relative coordinate setup

G52 X0Z0; 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

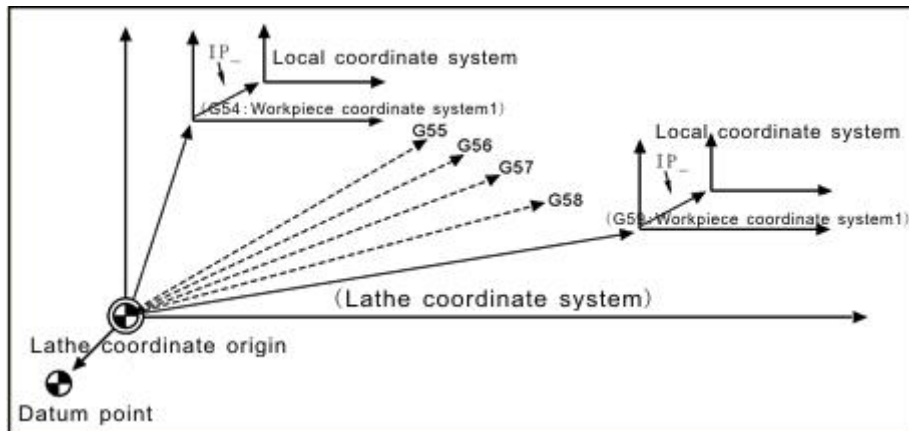
G185 X_Z_; setup all tool absolute coordinate

G185 U_W_; setup all tool relative coordinate

Use G52 to set sub coordinate system in lathe coordinate system and workpiece coordinate system. Set the origin of sub coordinate system in lathe coordinate system or use X- Z- to specify position in workpiece coordinate system.

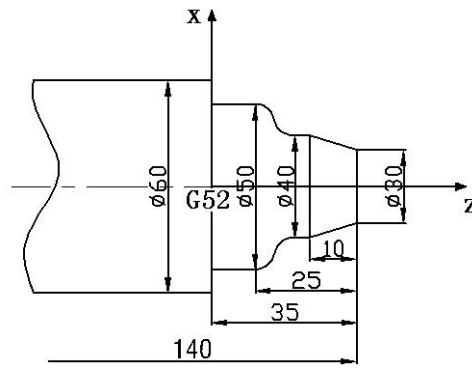
When setting local coordinate system, the motion with absolute mode is the coordinate value of local coordinate system. Use G52 to specify a new zero point which could change the position of local coordinate system. In order to cancel the local coordinate system and the specified coordinates in the workpiece coordinate system, should make the local coordinate system

Local coordinate system setting does not change the workpiece coordinate system and lathe coordinate system. When setting the workpiece coordinate system, if not specify all axis coordinate value, not to specify the coordinate value of the axis of the local coordinate system will not cancel but be remain unchanged. After the G52 program segment should specify motion instruction with absolute mode.



Part coordinate setup

For example:



```

N1 G00 X60 Z20
N2 G52 X0 Z-236      set part coordinate system;
N3 T0101             select tool number;
N4 M03 S800 M08      spindle rotate CW by 800rpm and coolant.
N5 G01 Z35 F100      move to end face Z35;
N6 X-1               cutting end face;
N7 X70               back tool;
N8 G71 U2 R1
N8 G71 P10 Q15 U0.5 W0.5 F150; excircle compound rough cutting loop;
N10 G01 X30
N11 X40 Z25
N12 Z20
N13 G02 X50 Z15 R5
N14 G03 X60 Z10 R5
N15 G01 Z0
N16 G00 X70          back tool rapidly;
N17 G52 X0 Z0        cancel part coordinate system;
N18 M05              stop spindle;
N19 M30              end program.

```

Example: processing five time on one workpiece.

```

G22 L5
X50 Z60 ; process workpiece;
.....
G52 W-50 ; Z axis offset by 50mm each time;
G800
G52 X0 Z0; cancel part coordinate system;
G0 X100 Z100
M02

```

(24) Back to starting point(G26、 G261-G268)

Back to start point of program(N00 line). G26 means both XZ back start point, speed is the same as G00 speed.

Format:

G26、 G261、 G262、 G263、 G264; XZ、 X、 Y、 Z、 A

Example:

N0000 G00 X120 Z300 ; move to workpiece face rapidly;

```

N0001 G01 X150 Z50 F160 ; feed to X150, Z50;
N0002 G26 ; X and Z back to start point simultaneously
N0003 M2 ; program end.

```

(25) Back to G25 point(G25、G61、G611-G618)

G25 remember the current X Z coordinate, G61 returns X Z coordinate point of G25, G611 returns X, G612 returns Y, G613 returns Z, G614 returns A, G25 is the same as G26 if not set.

Format:

G25 ;Save current coordinate

G61、G611、G612、G613、G614; XZ, X,Y,Z,A

For example:

```

N0000 G0 X20 Z80 ; rapidly move to X20, Z80.
N0001 G01 U5 W-16 F200 ; G01 interpolation, speed is 200.
N0002 W-100 ; cutting excircle, Z is 100.
N0003 G00 U10 ; X tool back by 10, to X35.
N0004 Z80 ; Z back to 80.
N0005 G25 ; memory coordinate X35, Z80.
N0006 G01 U10 W-30 ; processing to X45, Z50.
N0007 G0 X100 Z200 ; rapidly move to X100 Z200.
N0008 G61 ; back to X35, Z80.
N0009 M2 ; program end.

```

(26) Continue feed cutting(G60/G64)

Format: G60 ; cancel (mode)

G64 ; continue feed(mode, initial)

(27) Constant speed cutting(G96/G97)

Format: G96 S_ Constant speed cutting

G97 cancel

In the status of constant linear speed G96, S_ address word means linear speed (M / min)

After the cancellation of constant linear speed G97, S_ address word means spindle speed (R/M)

Press "absolute coordinate" to change the spindle speed in the status of constant linear speed G96.

Note:

G96: constant linear speed effective

G97: cancel the function of constant linear speed

G50 S: limit the maximum speed of spindle.

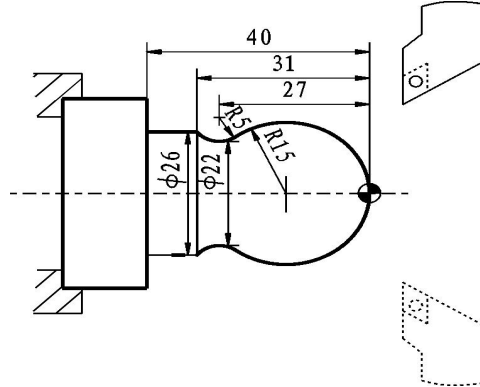
S:S value behind the G96 is the cutting constant linear speed, unit is m/min; S value behind the G97 is specifying spindle speed after cancel the constant linear speed; such as the default, is for the implementation of G96 Directive before spindle speed.

Note 1, use constant linear speed, the spindle must be able to change speed automatically. (such as: servo spindle, variable frequency spindle) set maximum limit speed in the system parameter.

Note 2, under control of the constant linear speed, the lower limit value of spindle speed in G96 can be set by No.35 parameter in speed parameter.

Note 3, spindle override couldn't work when processing in constant linear speed.

For example:



```

N1 T0102 X40 Z5           ;define tool set point
N2 M03 S400               ;spindle speed 400r/min
N3 G96 S80                 ;constant linear speed, speed is 80m/min
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              ;cancel constant linear speed, spindle 300r/min
N11 M30

```

(28) Feed mode(G98、 G99)

Format:

G98 feed method per minute instruction of G01, G02, G03. range:
0.01-15000mm/min;

G99 feed method per revolution instruction of G01, G02, G03. range:
0.01-15000mm/r;

When use G99, the spindle need to be equipped with encoder, or it will be standby state. And if spindle speed is slow(10r/min), processing may be not stable and balanced.

(29) Check skip(G31、 G311)

Format: G31 X_ Z_ F_ P_ ;No alarm

G311 X_ Z_ F_ P_ ;alarm

P:N line+(X00/X39+1000 or 2000), 1000 means availability skip,2000 mean 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. If X21 is

valid, then keeping alarm.

Note: input points X00-X39 can be checked in diagnosis interface.

(30) Work coordinate(G53/G54/G55/G56/G57/G58/G59)

Format: G53(G54/G55/G56/G57/G58/G59)

G53 machine coordinate

G54 workpiece coordinate 1

G55 workpiece coordinate 2

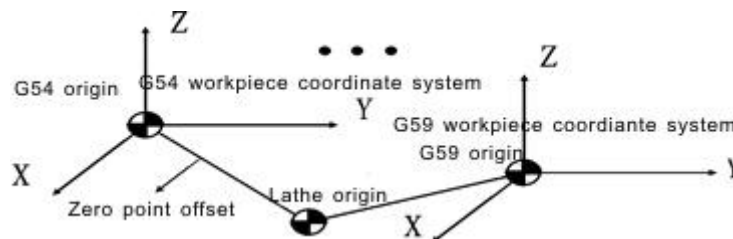
G56 workpiece coordinate 3

G57 workpiece coordinate 4

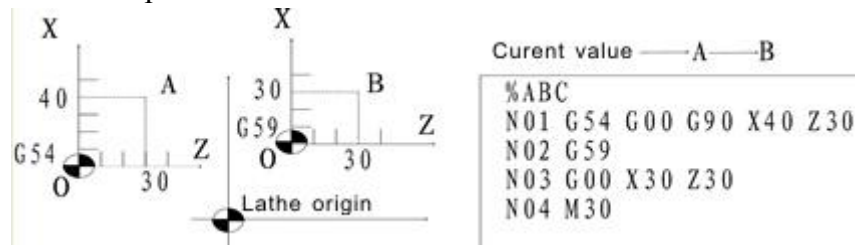
G58 workpiece coordinate 5

G59 workpiece coordinate 6

G53 is the machine coordinate which is define by home sensor or float home position. G54-G59 work piece coordinate offset can be set in parameter.



For example:



Note: 1. it is necessary to set original point of coordinate system in machine coordinate system before use these instructions.

2. it need to back reference point before use these instruction.

(31) Tool radius compensate G40、G41、G42

Format; G40 ; tool radius compensation cancellation, (mode,initial)

G41 ; tool radius compensation of cutter in the left of workpiece offset.(mode)

G42 ; tool radius compensation of cutter in the right of workpiece offset.(mode)

The left and right in G41, G42 are viewed from direction of cutting, tool lies in 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, system generates automatically transiting arc, allowing tool radius offset vector of last program segment transmit to that of the next program segment. Tool offset vector describes value and direction method of tool offset, its radius vector is tool radius. For arc, its direction is in radius

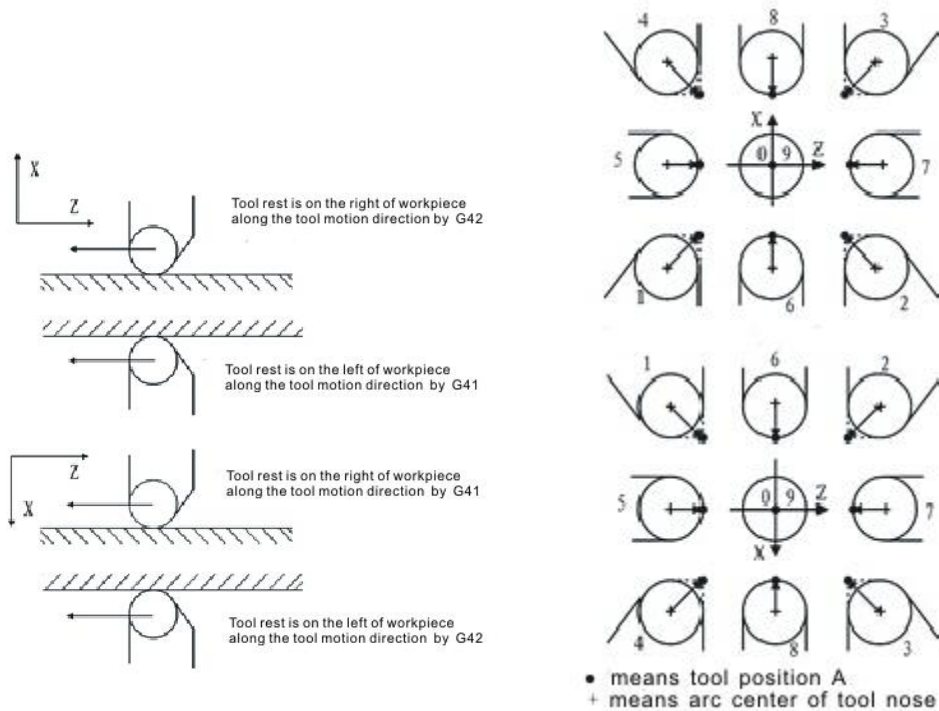
direction. For line, its direction is vertical to the line direction.

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.

G40: cancel tool endpoint compensation;

G41: left tool compensation(on the left side of tool moving direction);

G42:right tool compensation(on the right side of tool moving direction);

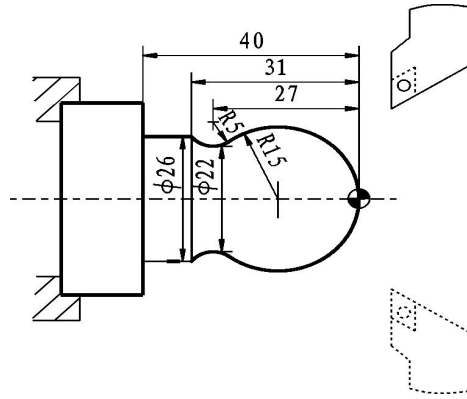


Left and right compensate

Note: (1) G41/G42 do not have corresponding parameter, the compensation No.(corresponding compensation value) is defined by T code, corresponding to tool offset compensation.

(2) the found and cancel of tool end point radius compensation only works with G00/G01, instead of G02/G03. tool endpoint radius compensation is in radius compensation of radeem interface, which defines tool arc radius; hypothetical tool endpoint number defines direction.

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

```

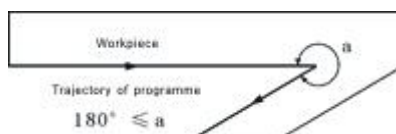
(32) Radius compensation of tool C

C means the system calculates the tool trajectory of radius compensation according to the last program line and the next program line.

1) Inside and outside

It calls inside when the included angle of tool trajectory is over 180 degrees which is built by two program segments, it calls outside when the included angle is between 0 and 180 degrees. As the follows:

Inside:



Outside:



2) Tool motion when starting

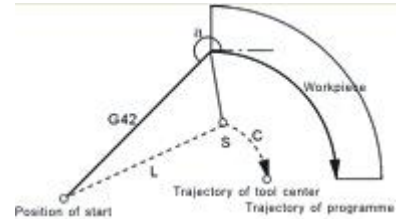
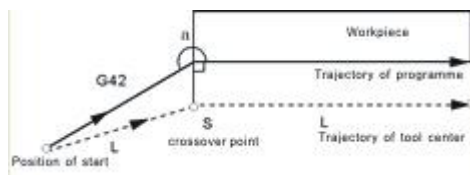
The radius compensation without tool builds tool radius compensation

(a) Tool motion around the inside corner ($\alpha \geq 180^\circ$)

The tool center will move to the tool vector radius vertex of the starting point in next program line.

Straight line->Straight line

Straight line->Arc

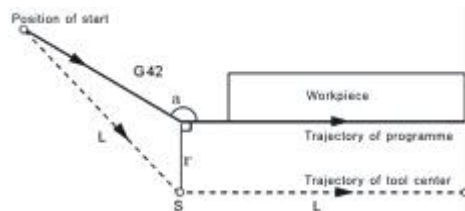


(b) The tool motion around the outside corner of obtuse angle ($90 \leq \alpha < 180$)

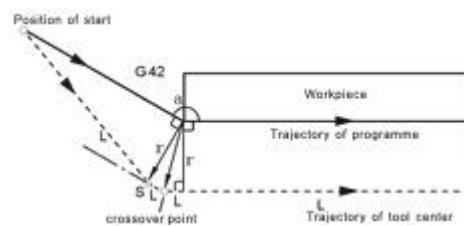
The tool center will move to the tool vector radius vertex of the end point in this program line.

Straight line->Straight line

A type

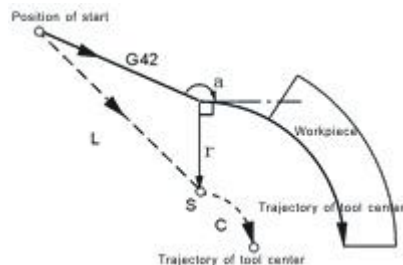


B type

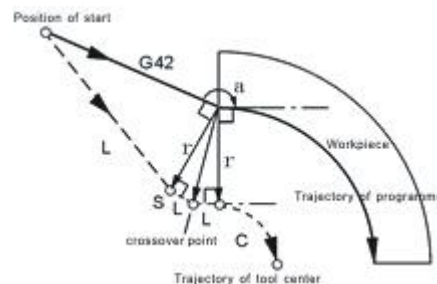


Straight line->Arc

A type



B type

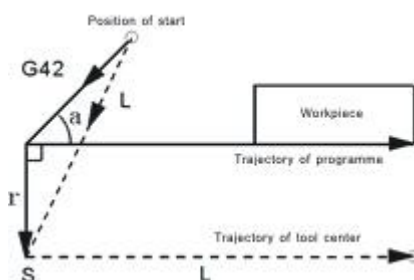


(c) The tool motion around the outside corner of acute angle ($\alpha < 90$)

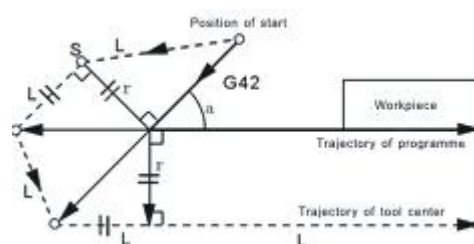
The tool center will move to the tool vector radius vertex of the end point in this program line.

Straight line->Straight line

A type



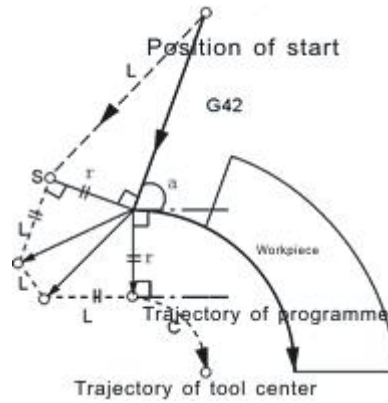
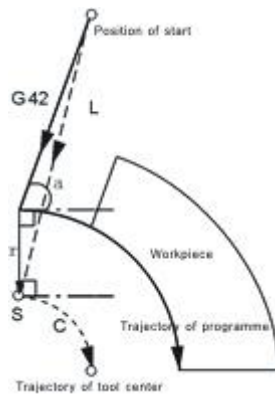
B type



Straight line->Arc

A type

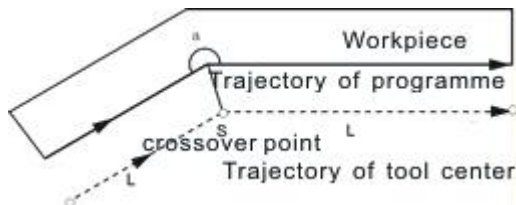
B type



2) Tool motion in offset mode

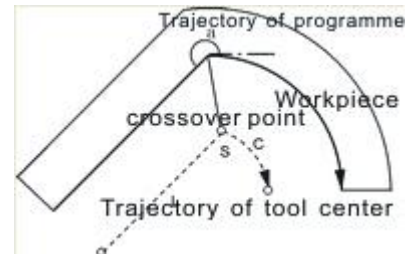
(a) Tool motion around the inside corner ($180 \leq \alpha$)

Straight line->Straight line

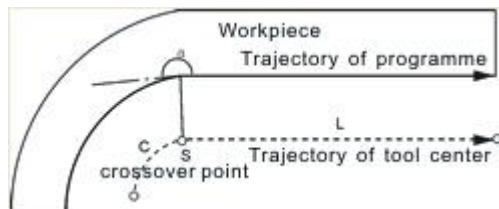


Arc-> Straight line

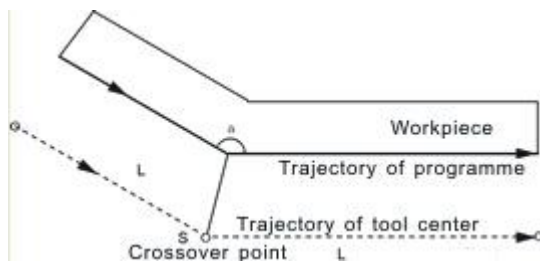
Straight line->Arc



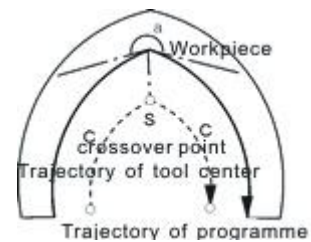
Arc->Arc



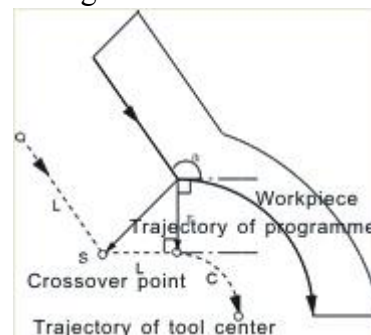
Straight line ->Straight line



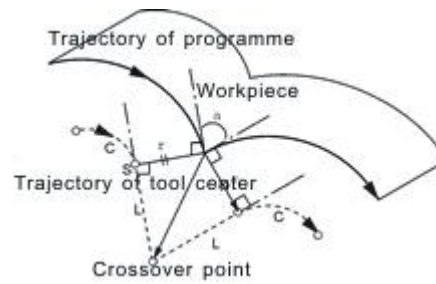
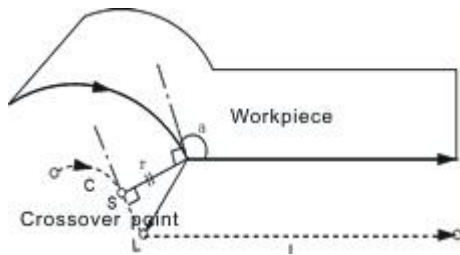
Arc-> Straight line



Straight line ->Arc



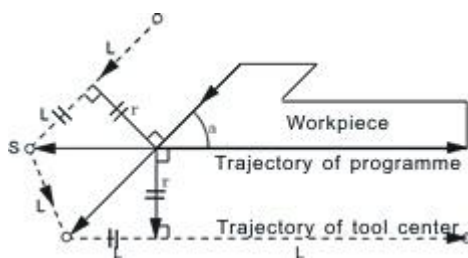
Arc->Arc



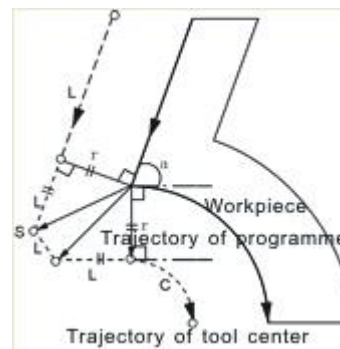
(c) The tool motion around the outside corner of acute angle ($\alpha < 90^\circ$)

Straight line → Straight line

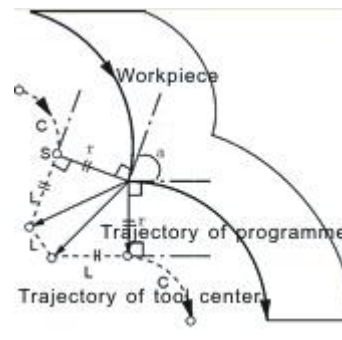
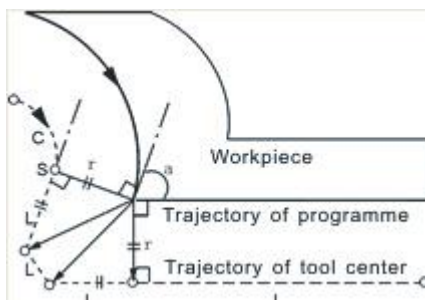
Straight line → Arc



Arc → Straight line



Arc → Arc



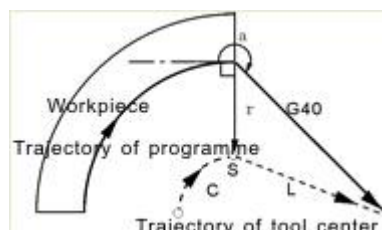
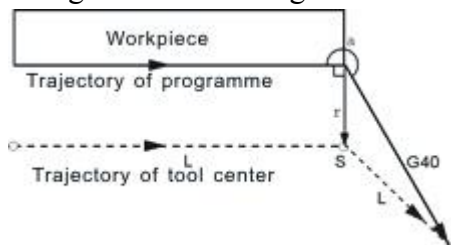
3) Tool motion in offset-cancel mode

(a) Tool motion around the inside corner ($180 \leq \alpha$)

The tool center will move to the tool vector radius vertex of the end point in this program line.

Straight line → Straight line

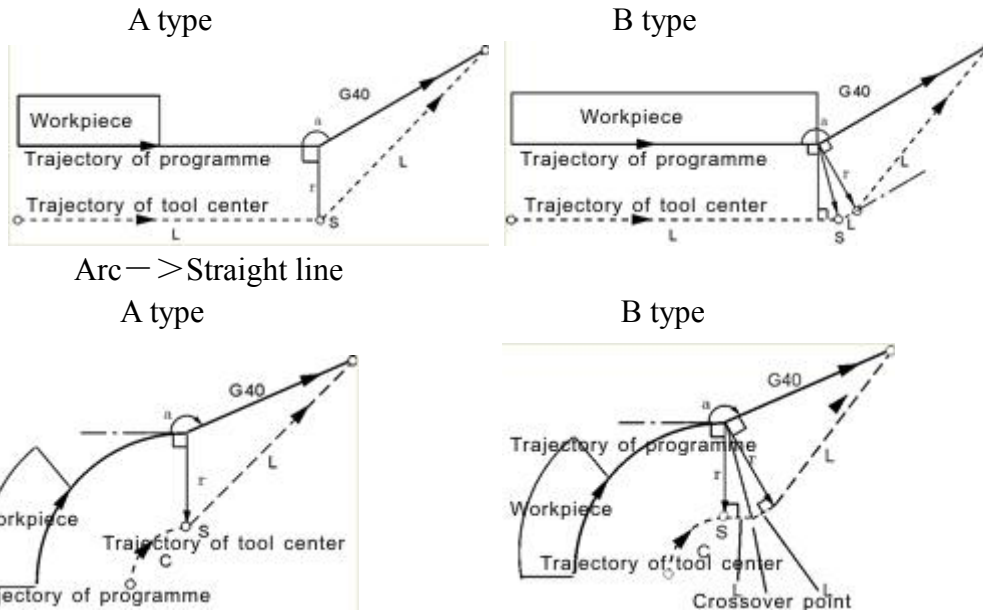
Arc → Straight line



(b) The tool motion around the outside corner of obtuse angle ($90 \leq \alpha < 180$)

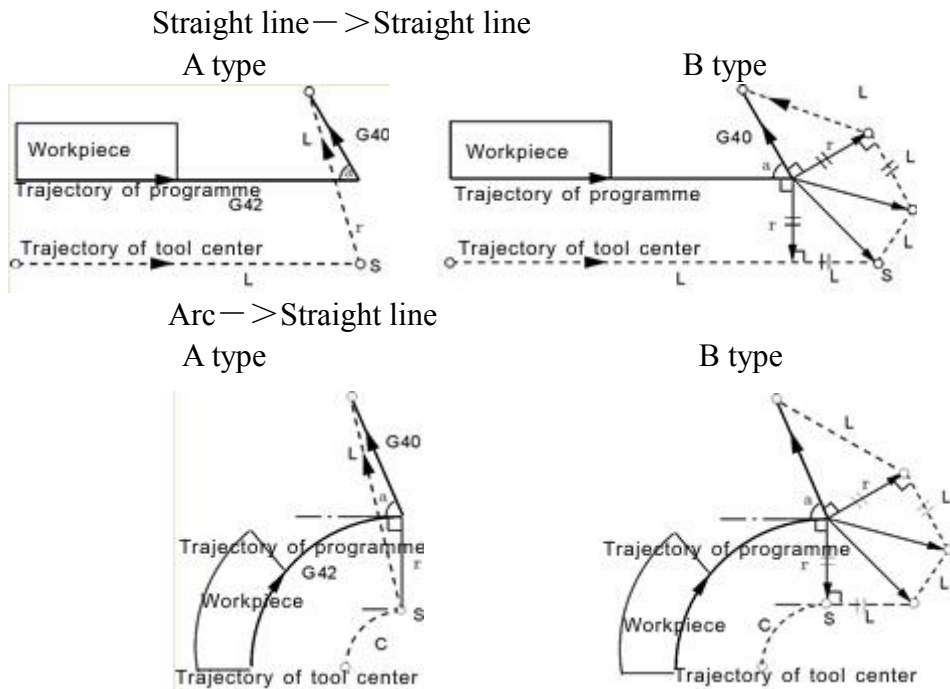
The tool center will move to the tool vector radius vertex of the starting point in next program line.

Straight line → Straight line



(c) The tool motion around the outside corner of acute angle ($\alpha < 90$)

The tool center will move to the tool vector radius vertex of the starting point in next program line.



(33) Automatic chamfer I and fillet R:

Format:

G01(G00) X(U) I: automatic chamfer, next segment including axis movements must be G01(G00) Z(W);

G01(G00) Z(W) I: automatic chamfer, next segment including axis movements must be G01(G00) X(U)。

G01(G00) X(U) R: automatic fillet, next segment including axis movements

must be G01(G00) Z(W);

G01(G00) Z(W) R: automatic fillet, next segment including axis movements must be G01(G00) X(U).

Attention:

1. address I and R must be radius mode. The movement of the current line and next line must be over width I of chamfer or radius R of fillet, or system will automatically modify the width I or radius R into the minimum distance of current linear and next linear movement.

2. two adjacent lines must be 90°.

Sample:

N0 M03S500

N1 G0 X0 Z0

N2 G01Z-5 F100

N3 X10I4

; charmfering 4x4

N4 X20R5

; fillet R5

N5 Z-20

N6 G26

N7 M30

(34) Polar coordinate program(G15/G16)

Polar coordinate instructions can be input by polar coordinate radius and angle, positive direction of angle is counter clockwise of Z axis positive direction, and negative direction is clockwise. Radius specifies with absolute instruction (Z), angle is with absolute instruction (X).

Format:

G15 ; cancel

G16 IP-(XZ) ; polar coordinate

Note:

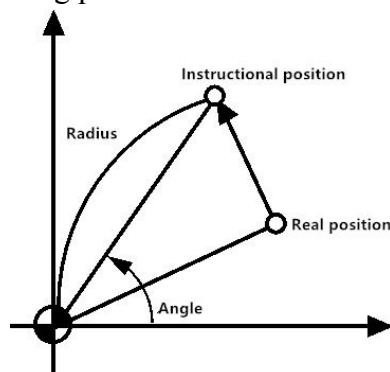
1. Z specifies the zero point of workpiece coordinate as the zero of polar coordinate system, from which measure the radius.

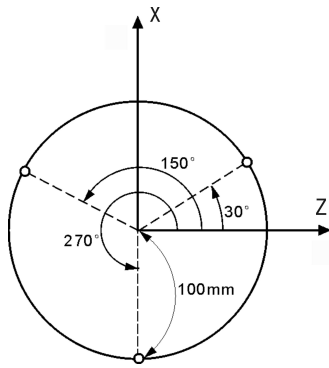
2. IP_ specifies the plane and the value of polar coordinate system by axis.

Z: radius of polar coordinate system;

X: angle of polar coordinate.

3. Z sets original of polar coordinate as original point of workpiec coordinate. As following picture.





For example:

N1 G16 X0 Z0 ; sets original of polar coordinate as original point of workpiece coordinate;

N2 G01 X30.0 Z100.0 F200.0; specify distance 100mm, angle 30°;

N3 X150.0 ; specify distance 100mm, angle 150°;

N4 X270.0 ; specify distance 100mm, angle 270°;

N5 G15 ; cancel polar coordinate instruction;

N6 M02

(35) Polar coordinate interpolation(G12.1/G13.1)

Converting shape controlling from instruction in rectangular coordinate into the movement of one linear axis(tool movement) and one rotary axis(workpiece rotary movement).

Format:

G12.1; start Polar coordinate interpolation;

G13.1; cancel Polar coordinate interpolation;

Note: G12.1, G13.1 are specified by single segment.

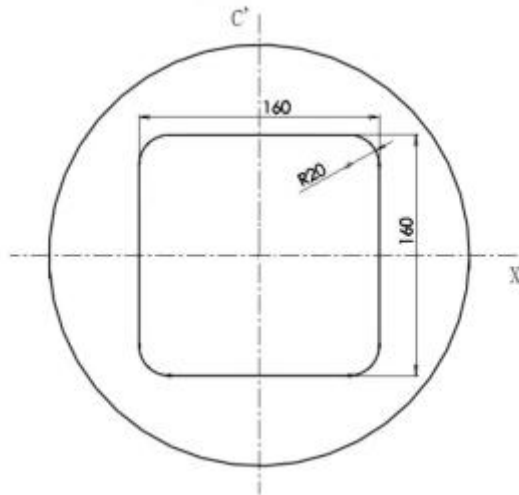
After enable the polar coordinate, you can instruct linear interpolation or arc interpolation in rectangular coordinate, containing of linear axis and rotary axis.

G12.1 starts polar coordinate system and choose a plane where polar coordinate interpolation is finished.

Process: polar coordinate interpolation based on X axis(linear) and C axis(rotary).

X axis is programmed by diameter or radius, C axis is programmed by radius(program unit: mm, display unit: angle)

Sample:



N10 T0202

...

N100 G00 X150 C0 Z0;

N110 G12.1;

N120 G01 X80 F200;

N130 C20;

N140 G03 X40 C40 R20;

N150 G01 X-40;

N160 G03 X-80 C20 R20;

N170 G01 C-20;

N180 G03 X-40 C-20 R20;

N190 G01 X40;

N200 G03 X80 C-20 R20;

N210 G01 C0;

N220 X150.0;

N230 G13.1;

N240 Z100.0 C0

...

N500 M30

Note 1: when controller power on or reset, polar coordinate interpolation will be canceled(G13.1); G12.1 and G13.1 are mode code.

Note 2: execution of G12.1 will be converted into G17 plane; execution of G13.1 will back into G18 plane;

Note 3: during polar coordinate interpolation, program is coded with rectangular coordinate instruction in polar coordinate system. No matter the linear axis in plane use diameter or radius, the rotary axis is programmed with radius.

Note 4: the available G codes in polar coordinate interpolation:

G01: linear interpolation;

G02、G03: arc interpolation;

G04: pause;

G40、G41、G42: tool endpoint compensation;

Note 5: during polar coordinate interpolation, speed of F code is the speed tangency speed with polar coordinate interpolation plane(rectangular coordinate system);

Note 6: the arc interpolation in polar interpolation plane will specify arc radius according to G17 interpolation plane, linear axis X use I, rotary axis C use J. also can use R code;

Note 7: in G12.1, coordinate system cannot be changed.

Note 8: in tool endpoint interpolation mode(G41 or G42), polar coordinate interpolation cannot be started or canceled; only if in G40 mode, G12.1 or G13.1 can work.

Note 9: in polar coordinate interpolation, when tool move near the center pint of workpiece, component feed speed of C axis will rise, once over the maximum cutting speed of C axis, controller will alarm!

Note 10: during polar coordinate interpolation, program is rectangular code in polar coordinate plane. Axis address of rotary axis is address of the second axis(virtual axis) in plane. When instructing G12,1, tool position of polar coordinate interpolation starts from 0 degree. So before polar interpolation, spindle should be orientated as 0 degree. And before cancel the polar interpolation, C axis should be positioned at zero point of coordinate.

Note 11: in polar coordinate interpolation , the current position is real coordinate. But the balance distance in program segment will display according to polar coordinate interpolation plane(rectangular coordinate);

Note 12: during polar coordinate interpolation, please do not switch spindle gear.

(36) Metric system and British system (G20/G21), absolute program and incremental program(G990/G991)

Format:

G990 ; XZYAC is absolute program;
G991 ; XZYAC is incremental program;
G20 ; inch input;
G21 ; mm input;

Note:The G code must be compiled in the beginning of the program, using separate program segment to specify before setting the coordinate system.Switch the unit of input data into minimal inch or millimeter after G code of switching inch or millimeter specifying,the angle of data input unit keeps unchanged, change the units of value as follows after switching the inch or millimeter:

- The feeding speed is specified by F code
- Position instruction
- Offset value of workpiece zero point
- Compensation value of tool
- The unit of manual pulse generator
- The distance in incremental feeding

Specification of unit under G20/G21(it is necessary to reset the unit in parameter):

- 1) The minimum unit of British system G20 is 0.0001inch, the minimum unit of Metric system is 0.001mm.
- 2) Under British system G20, unit of all instructions related to position is inch;
- 3) Under British system G20, unit of feeding speed F instruction is inch/min;
- 4) Under British system G20, unit of tool compensation and offset of workpiece coordinate system G54-G59 is inch;
- 5) Under British system G20, unit of parameters related to distance(such as soft limit and compensation) change from mm of Metric system into 0.1 inch or from um into 0.001inch;
- 6) Under British system G20, unit of parameters related to speed change from mm/min into 0.1inch/min ;
- 7) Under British system G20, unit of parameters related to accelerated speed change from mm/min/s into 0.1inch/min/s;
- 8) Under British system G20, handwheel incremental value are 0.0001inch, 0.001inch, 0.01 inch;
- 9) Under British system G20,if pulse equivalent changes, system will adjust automatically, electronic gear will accords to metric system;
- 10) Under British system G20, coordinate displays four numbers behind the decimal point;
- 11) G20/G21 should be written in the beginning of program, instead of middle of program;
- 12) Under British system G20, rotary axis rotating 1 degree accords to 0.1 inch;
- 13) two systems can be converted by executing G20/G21 in MDI, and restating controller.
- 14) It is necessary to set tool after conversion between G20 and G21;
- 15) As to system prompt information, you just need to modify the parameter configuration file cncsystemen. For example, the parameter about speed: change mm/min into 0.1inch/min, parameter about position:change mm or degree into 0.1 inch, change um into 0.0001 inch.

(37) M instruction

Usage	instruction	Function	Initial	mode	Note
-------	-------------	----------	---------	------	------

Spindle	M03 M04 M05 M203 M204 M205 M881 M882	Spindle on CW Spindle on CCW Spindle brake M7053/M7054 Pxxxx: Spindle stop after rotate CW/CCW a while, the time is specified by P, unit:ms Example: M7053 P2000; means spindle rotate CW 2s and then stop C axis stop, output M61, check M22 A axis stop, output M63, check M24	√	√ √ √	M203 M204 M205 The second spindle rotate CW, CCW, stop
Cooling	M08 M09	Coolant on Coolant off	√	√ √	
Chuck	M10 M11	Chuck tightens Chuck loosens	√	√ √	
Tail stock	M79 M78	Tighten Loose	√	√ √	Control M79
Lubrication	M32 M33	Lubrication on Lubrication off	√	√ √	Control M32
Huff	M59 M58	Huff on Huff off	√	√ √	Control M59
User-defined output	M61 M60	user-define 1 open user-define 1 close		√ √	Control M61
	M63 M62	user-define 2 open user-define 2 close		√ √	Control M63

	M65 M64	user-define 3 open user-define 3 close		√ √	Control M65
	M67 M66	user-define 4 open user-define 4 close		√ √	Control M67
	M69 M68	user-define 5 open user-define 5 close		√ √	Control M69
	M71 M70	user-define 6 open user-define 6 close		√ √	Control M71
	M73 M72	user-define 7 open user-define 7 close		√ √	Control M73
	M75 M74	The spindle servo select the position controlling mode when C axis is output pulse signal, close M75 signal when system starting M03/M04		√ √	Control M75
user-define d input	M12 M13	Check M12 valid Check M12 invalid		√ √	Conditional skip, example: M20 P120 means if M12 is valid, program skips to 120 th line.
	M14 M15	Check M14 valid Check M14 invalid		√ √	
	M16 M17	Check M16 valid Check M16 invalid		√ √	
	M18 M19	Check M18 valid Check M18 invalid		√ √	
	M28 M29	Check M28 valid Check M28 invalid		√ √	
	M22 M23	Check M22 valid Check M22 invalid		√ √	
Spindle Gear	M41 M42 M43 M44	first gear second gear third gear fourth gear			Input S01, S02, S03, S04 and modify the analog voltage of spindle.
Subprogram	M97 M98 M99	jump call return			L=1-99 P is the line number of transferring program

Program	M87 M188 M187 M00 M01 M02 M30 M20	Part number plus 1 Keep feeding and not back home Keep feeding and back home pause Conditional pause, check M22 Program end M05、 M09 end Loop go start			When other parameter P10=0 is set not to autotomatical plus 1, instruction M87 to make workpiece number plus 1
Read the position of absolute motor	M500 M501 M502 M503 M504	M500: read absolute motor position of all the feeding axis and reset the current lathe coordinate. M501: read absolute motor position of X axis and reset the current lathe coordinate. M502: read absolute motor position of Y axis and reset the current lathe coordinate. M503: read absolute motor position of Z axis and reset the current lathe coordinate. M504: read absolute motor position of the fourth axis and reset the current lathe coordinate.			
Clear workpiece coordinate	M315 M317 M318 M319 M320	M315:Clear workpiece coordinate of A axis M317:Clear workpiece coordinate of X axis M318:Clear workpiece coordinate of Y axis M319:Clear workpiece coordinate of Z axis M320:Clear workpiece coordinate of all axis			

Clear lathe coordinate	M415 M417 M418 M419 M420	M415:Clear lathe coordinate of A axis M417:Clear lathe coordinate of X axis M418:Clear lathe coordinate of Y axis M419:Clear lathe coordinate of Z axis M420:Clear lathe coordinate of all axis			
Ratio	M100 M101	Ratio invalid, fixed ratio is 100%; Ratio valid.			
Chip breaking	M114 M115	M114 Lxx(Qxx) Pxx Fxx M115 cancel			

All of M code input and output are low-level effective.

(38) Chips breaking/shaping machine specific instruction(M114/M115)

M114 Lxx(Qxx) Pxx Fxx ; chip breaking instruction, mode instruction

M115 ; chip breaking cancel instruction, mode instruction.

When M114 includes P code, it mean lathe chip breaking instruction, value behind P is pause time, unit:ms.

When M114 does not includes P code, it means shaping machine planing instruction, at this time, it will not pause, but execute ProgramUser6 file.

In chip breaking mode, M114 only works with G01/G02/G03 and loop code, ineffective for tapping, threading and G00.

Note:

1) in chip breaking mode, G01/G02/G03 and loop code will pause time which is defined by P after run L time(or Q distance);

2) L define pause time by the F speed afterwards,unit:ms

Q define pause distance by the F speed afterwards, unit:mm.

3) F define feeding speed, unit: mm/min.

4) unit of Q in british system is inch, F speed is in/min.

(39) Call program (M97/M98/M99)

Unconditional skip

M97 P: unconditional jump to P-word line;

Call sub-program:

In the controller, subprogram should be an independent program.

M98 P L: unconditional call subprogram. P specify route and name of subprogram, L is the address of calling times.

M98 can be omitted, format is PP file name, file name can be hidden file, but the head character must be "HIDEFILE". for example: file "HIDEFILE01" will not be shown in program section, it can be called by M98/G65 PHIDEFILE01or M98/G65

P*01 or PP*01 or PPHIDEFILE01.

For example: Psub/1390 means subprogram is tmp/NC/sub/1390.

Attention:

1. tmp/NC/ is default route of controller program, sub is one of folders under it.
2. subprogram should be an independent program.
3. the method calling which happened between both main program and subprogram in USB disc. P [or P];

Sample: M98 P[A1234 means call subprogram A1234 in USB;

M98 P]SS12 means call subprogram SS12 in USB;

PP[FFDE means call subprogram FFDE in USB;

If call program under folder, it is necessary to edit route.

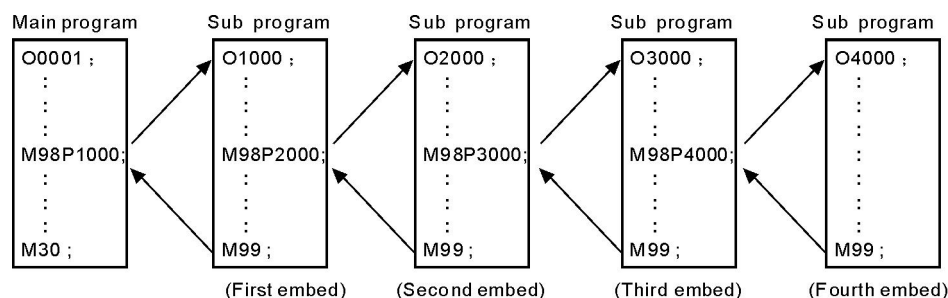
There must be a space before call times address L. the subprogram is finished, then back to next segment of main program. If program includes fixed sequence or repeated shape, then this kind of sequence or shape can be edited as a subprogram to simplify program. Main program can call subprogram, and subprogram can also call another subprogram.

M99: finish subprogram and back, subprogram finishing must use this.

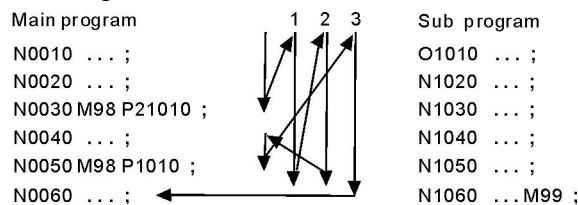
Attention:

- 1) M99 is identical with M02 in main program;
- 1) M99 with P is identical with M97 in main program;
- 1) M99 is in the next line of calling when back to main program;
- 1) M99 with P is in the P line when back to main program;

The Sub-program can embedded call under four levels as follows:



For example



The maximum call times is 9999.

(40) Conditional wait or skip

N code can be used to detect outer input signal as condition:

Conditional wait:

M12 M13 used to detect input status of signal M12, M12 in program means detect M12, if valid then execute next line. M13 means detect M12 signal, if invalid then execute next line. The instruction need to be edited in an isolated line.

M14 M15 used to detect input status of signal M14, M14 in program means detect M14, if valid then execute next line. M15 means detect M14 signal, if invalid then execute next line. The instruction need to be edited in an isolated line.

M16 M17 used to detect input status of signal M16, M16 in program means detect M16, if valid then execute next line. M17 means detect M16 signal, if invalid then execute next line. The instruction need to be edited in an isolated line.

M18 M19 used to detect input status of signal M18, M18 in program means detect M18, if valid then execute next line. M19 means detect M18 signal, if invalid then execute next line. The instruction need to be edited in an isolated line.

M22 M23 used to detect input status of signal M22, M22 in program means detect M22, if valid then execute next line. M23 means detect M22 signal, if invalid then execute next line. The instruction need to be edited in an isolated line.

M24 M25 used to detect input status of signal M24, M24 in program means detect M24, if valid then execute next line. M25 means detect M24 signal, if invalid then execute next line. The instruction need to be edited in an isolated line.

M28 M29 used to detect input status of signal M28, M28 in program means detect M28, if valid then execute next line. M29 means detect M28 signal, if invalid then execute next line. The instruction need to be edited in an isolated line.

Conditional skip:

M12/M13/M14/M15/M16/M17/M22/M23/M24/M25/M28/M29 add Pxxxx (line number of program), means if meet condition then skip to specified line, otherwise execute next line.

Example: M14 P0120

When program execute to this line, controller will detect M14 input signal, if M14 is valid, program will skip to 120th line(marked line), if M14 is invalid, then execute next line.

(41) T tool function

Format	function
Tab	a: Tool number, b: compensate number a=0 means not change tool B=0 means cancel tool compensation, and display machine coordinate. a=0~99, b=0~99.

For example: T0204 ; change No.2 tool and No.4 tool compensation;

T0300 ; change No.3 tool and display machine coordinate;

T0004 ; not change tool and change No.4 tool compensation.

When install electronic turret, each tool position can equip different tools, then adjust different different tool compensation. For example, in four-tool turret, and install 2 tools in position 3(No.3 tool compensation, No.5 tool compensation).

Format:

N0000 T0101

N0001 G0 X30 Z500

N0002 T0303 ; change No.3 tool and No.3 tool compensation.

N0003 G00 X50

N0004 T0505 ; No.3 tool and No.5 tool compensation.
N0005 M02

Attention:

1. Txxxx, the first two number are tool number, last two number are tool compensation number. Range of tool number is 01-99, tool compensation can be called in any one.

2. when execute tool compensation, machine does not move, just calculate the difference and update coordinate. If want to display coordinate movement according to tool compensation, please programme with absolute instruction.

3. after power on, controller will execute the tool number before power off, and execute default compensation of default tool.

4. change tool in manual mode, the number of compensation will accords to tool number. For example, No.6 tool will execute No.6 compensation.

(42) S、SS Spindle speed

There are two ways to control spindle:

The first SP use "S", and has two ways to control,

(1) output step speed change with four bit code, controlled by M41-M44, corresponding output S01-S04. Use axis parameter P50,51,P52,P53,P54 to set the max speed.

(2) four gears with stepless speed change, controlled by M41-M44, corresponding output S01-S04. Use axis parameter P36,P37,P38,P39 to set the max speed. Use speed parameter P50,51,P52,P53,P54 to set change method.

Stepless speed change ranges 0-99999, output 0-10V frequency conversion voltage. Output voltage is specified Sx override/max speed.

The second SP use "SS", speed parameter P40 control the highest speed, output 0-10V frequency conversion voltage.

(43) F feed speed

F use for G01、G02、G03 etc. feed speed.

G98: range F0.01-20000mm/min, G99: range 0.001-500mm/r.

Feed speed is Fx speed override, F is mode instruction.

(1) G98 feed speed per minutes. mm/min;

(2) G99 feed speed per turn. Mm/r.

If F instruction and movement instruction is in the same line, it will execute F instruction firstly.

Attention: G99 instructions requires spindle with encoder, or controller will be under waiting.

(44) Macro program instruction(G65, G66, G67)

1.Input instruction: WAT

Waiting for the input port X valid or invalid instruction

Format: WAT+ (-) X

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

"-" means the input is invalid;

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

2.The output instruction: OUT

Set the output port Y is valid or invalid instruction

Format: OUT +(-)Y

Note: "+" means the output is effective;

"-" means the output is invalid;

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

3. Variable and assignment: =

1) #0--#20 local variable: local variables only can be used to store data in macro program, such as a result of operation, when power is off, the local variables are initialized to the empty. The argument assignment to the local variable when calling the macro program.

2) #21--#600 global variables: The meanings are the same in different macro program.

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

3) #1000-- system variable: the system variables are used to change various data when reading the running CNC. For example, the current position and the compensation of tool.

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

The global variable available for user are: #31-#99, #205-#379, #391-#999, the rest are occupied by system.

#161-#165 can used to display or record XYZAB coordinate in G31/G311.

#201: repeat times of G76(method 1).

#202: tool offset value in Z direction when G76 repeats. Unit:mm,(method 1).

#203: repeat times of G76(method 2).

#204: tool offset value in Z direction when G76 repeats. Unit:mm,(method 2).

Attention: macro program can not use loop code: G70,G71,G72,G73,G92,G76.

4) The macro variables #1001--#1099 corresponds the X axis offset value of lathe T1--T99(Unit: micron)

The macro variables #1401--#1499 corresponds the Z axis offset value of lathe T1--T99(Unit: micron)

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

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

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

5)The I/O variables:

#1800: X00-X07 (D0-D7)

#1801: X08-X15 (D0-D7)

#1802: X16-X23 (D0-D7)

#1802: X16-X23 (D0-D7)

#1803: X24-X31 (D0-D7)

#1804: X32-X39 (D0-D7)

#1805: X40-X47 (D0-D7)

#1806: X60-X67 (D0-D7)

#1808: Y00-Y15 (D0-D15)

#1809: Y16-Y31 (D0-D15)

Format:#i=Expression

6) save the current global macro variable with format of customer program
FILEMS(AABBCC) or FILEMS[AABBCC]

Save #21-#999 into program AABBCC file;

If want to read these variable, just need to execute the file or call the file as a subprogram.

4.The arithmetic and logic operation

Table:

Function	Format	Note
Definition	#i = #j	
Addition	#i = #j + #k ;	
Subtraction	#i = #j - #k ;	
Multiplication	#i = #j * #k ;	
Division	#i = #j / #k ;	
Sin	#i = SIN(#j) ;	90.5 degrees mean 90 degrees 30 minutes
Asin	#i = ASIN(#j);	
Cos	#i = COS(#j) ;	
Acos	#i = ACOS(#j);	
Tan	#i = TAN(#j);	
Atan	#i = ATAN(#j);	
Square root	#i = SQRT(#j);	
Absolute value	#i = ABS(#j) ;	
Rounding off	#i= ROUND(#j);	
Round down	#i = FIX(#j);	
Round up	#i = FUP(#j);	
Natural logarithm	#i = LN(#j);	
Exponential function	#i = EXP(#j);	
Or	#i = #j OR #k ;	Executing with binary system
Exclusive or	#i = #j XOR #k ;	
And	#i = #j AND #k ;	

5. Unconditional transfer: GOTO N

Transfer to the program line with sequence number appears error when specifying beyond the 1-99999, could use expression to specify the sequence number.

For example: GOTO 5, GOTO#100

6.Conditional transfer: IF (Conditional expression) GOTO or THEN

If the conditional expression specified met, execute this segment; if the conditional expression specified does not meet, execute the next segment.

For example: IF (#100 EQ 2) THEN #100=5

IF (#101 GT 2) GOTO 6

IF (#101 GT 2) GOTO 6

Operation meaning:

EQ equal

NE not equal

GT greater than >

GE greater than or equal

LT less than <

LE less than or equal

7. Cycle: WHILE (conditional expression) DO 1, 2, 3

Specifies a conditional expression in front of WHILE. When the specified conditions are met, execute the program between DO and END. Otherwise, turn to the program line after END. Cycle of the embed is 3 at the most.

For example: WHILE (#100 LT 3) DO 1

```
.....  
WHILE (#103 EQ 5) DO 2  
  
.....  
WHILE (#200 GE 20) DO 3  
  
.....  
END 3  
  
.....  
END 2  
  
.....  
END 1
```

8. Non-mode to call macro program: G65

Format: G65 P- L- <A-B-C-..... Argument passing data >

P is the name of macro program, L is the calling times, A B C are argument, the name of argument as follows:

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

Special attention: The address G、L、N、Q、P can't be used in argument.

For example:

```
Main program:9000  
G00 X0 Z0  
G65 P8000 L1 A5 B6  
G0 X0 Z0  
M30  
Macro program:8000  
N1 #2=#0+#1  
N2 IF (#2 EQ 10) GOTO 4  
N3 G00 X#2  
N4 G00 Z#1  
N5 M99 ; Return
```

9. Mode to call macro program: G66 G67

G67 instruction is to cancel G66 instruction. The format is the same as G65.

For example:

```
Main program:9000
G00 X0 Z0
G66 P8000 L2 A5 B6
A8 B1
A9 B10
G67
M30
Macro program:8000
N1 #2=#0+#1
N2 IF (#2 EQ 10) GOTO 4
N3 GOO X#2
N4 G00 Z#1
N5 M99 ; Return
```

10.Information prompt dialog

Format: MSG(parameter) or MSG[parameter]; parameter is information character string, pause.

Note: the instruction also can work in normal NC program(none macro program).

Format: STAF(parameter) or STAF[parameter]; parameter is information character string, no pause.

11.Generate process program automatically

1>, the instruction of found and open file:

FILEON(parameter) or FILEON[parameter]

Example:

FILEON(AABBCC) or FILEON[AABBCC]

means found and open file AABBCC

2>,the instruction of close file:

FILECE means close the current file, if without the instruction, controller will close the file automatically after the program finish.

3>,write character string into the current opening file:

FILEWD (parameter) or FILEWD [parameter]

Example:

FILEWD (G54G0X0Z0) or FILEWD [G54G0X0Z0]

means edit character string into the current opening file G54G0X0Z0

4>, write absolute coordinate of current feed axis into the file:

FILEWC

Sample program:

```
G0X0Z0
FILEON[AABBCC]
FILEWD [G54G0X0Z0]
G1X45Z89
FILEWC
G1X99Z76
FILEWC
```

FILECE

After finish the program, it will generate a file AABBBCC in program directory, contents is as following:

G54G0X0Z0

X45Z89

X99Z76

(45) User-defined macro instruction(G120-G160,M880-M889)

Every user-defined G code is corresponding to a macro program ProgramGxxx, the M code is corresponding to a macro program of ProgramUser0 --ProgramUser9, the user cannot programme the macro program in NC system, must edit the macro code in the computer, and then copy into the system.

For example, defines the G152 function: the arc model porous drilling cycle. (must copy the macro program ProgramG152 into system).

Format:G152 Xx Yy Zz Rr Ii Aa Bb Hh Ff;

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

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

Z: Hole depth

R: Approaching fast to the point coordinate

F: Cutting feed speed

I: Radius

A: The angle of the first hole

B: Incremental angle specify(CW when negative)

H: quantity of hole

Macro program ProgramG152 as follows:

#80=#0

#81=#1

#82=#2

#83=#3

#84=#4

#85=#5

#86=#6

#87=#7

#88=#8

#89=#9

#90=#10

#91=#11

#92=#12

#93=#13

#94=#14

#95=#15

#96=#16

#97=#17

#98=#18

#99=#19

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

(46) Composite function of turning and milling.

The method of performing milling in lathe controller:

1. parameter: enable the third axis, and set it as Y axis, the program method set as radius.
2. relative instruction: except following instruction, the other instructions refer to milling controller manual.

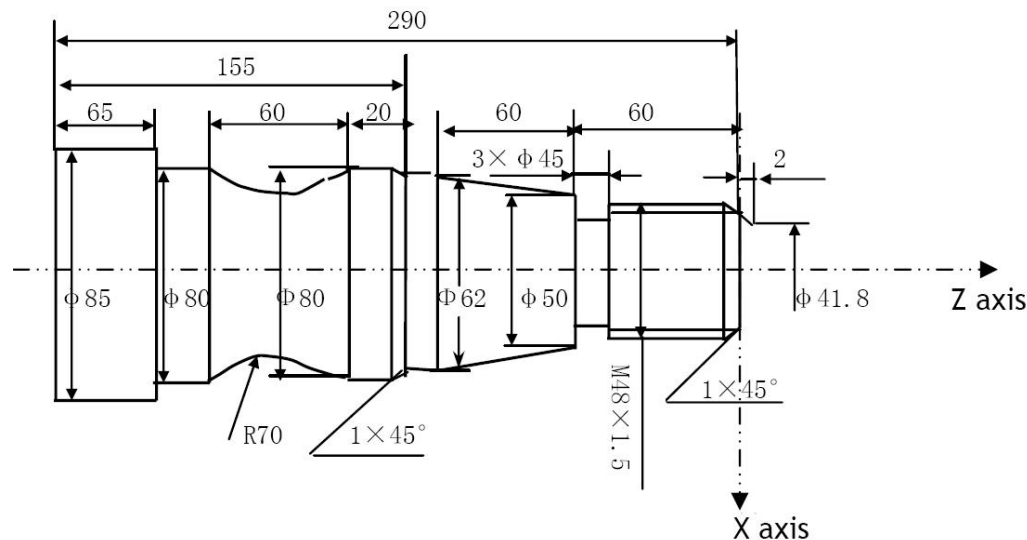
G990 accords to G90 in milling controller;
G991 accords to G91 in milling controller;
G994 accords to G94 in milling controller;
G995 accords to G95 in milling controller;
G998 accords to G98 in milling controller;
G999 accords to G99 in milling controller;
G973 accords to G73 in milling controller;
G974 accords to G74 in milling controller;
G976 accords to G76 in milling controller;
G981 accords to G81 in milling controller;
G982 accords to G82 in milling controller;
G983 accords to G83 in milling controller;
G984 accords to G84 in milling controller;
G985 accords to G85 in milling controller;
G986 accords to G86 in milling controller;
G987 accords to G87 in milling controller;
G989 accords to G89 in milling controller.

4.5 Program Sample

In the actual programming, must according to the drawing and processing requirement to select the correct install folder mode and suitable tool, and combined with the actual working performance of lathe to select the right cutting allowance, for example:

Example 1: The tool is:

T01 cylindrical cutting tool; T02 cutting groove, tool width 3mm; T03 thread tool with 60 degree angle



Program as follows:

N10 M03 S1000;	Start spindle
N20 T0101;	Choose the first tool and execute the first redeem
N30 G00 X41.8 Z2 M08;	Move fast to the cutting point, cutting fluid is on
N40 G01 X48 Z-1 F100;	Chamfer
N50 Z-60;	Fine machining for thread
N60 X50;	Tool is backing
N70 X62 W-60;	Fine machining in cone
N80 W-15;	Fine machining in $\phi 62$ mm excircle
N90 X78;	Tool is backing
N100 X80 W-1;	Chamfer
N110 W-19;	Fine machining in $\phi 80$ mm excircle
N120 G02 X80 W-60 R70;	Fine machining in arc (I63.25 K-30)
N130 G01 Z-225;	Fine machining in $\phi 80$ mm excircle
N140 X85;	Tool is backing
N150 Z-290;	Fine machining in $\phi 85$ mm excircle
N160 X90 M09;	Tool is backing, cutting fluid is off
N170 G00 X150 Z50;	Move fast to the point of changing tool
N180 T0202;	Change tool and set the No.2 redeem
N190 M03 S800;	Change speed of spindle

N200 G00 X51 Z-60 M08; Move fast to the processing point, use the left point of tool to redeem
 N210 G01 X45 F90; Cutting Φ 45mm groove
 N220 G00 X51; Tool is backing
 N230 X150 Z50 M09; Return to the point of backing tool, cutting fluid is off
 N240 T0303; Change tool and set the redeem
 N250 M03 S1500; Change speed of spindle
 N260 G00 X62 Z6 M08; Move fast to the processing point, cutting fluid is on
 N270 G92 X47.54 Z-58 F1.5; Cutting thread is cycle
 N280 X46.94;
 N290 X46.54;
 N300 X46.38;
 N310 G00 X150 Z50 M09; Return to the point of start cutting, cutting fluid is off
 N320 T0300; Cancel redeem
 N330 M05; Stop spindle
 N340 M30; Program is over

4.6 Usage for grinder

Use for excircle grinder with active measure and control instrument to process

1、Face grinding method

T0101

M60 ; Use M60 to choose the mode of face measurement

M03 ; Start the spindle of emery cutter

M63 ; To measure the start to the measuring position, the system

input M63 single when the lathe equipping this equipment

G04 X1 ; Delay 1 second

G00 X0 Z1 ; To the nearby face measurement

G31 W-3 F100 P1031 ; Rough machining in face enough M16

G31 W-1 F10 P1028 ; Fine machining in face enough M18

G0 Z1

M62 ; Start the equipment of face measurement

M02

2、Excircle grinding mode T0101

M61 ; Use M61 to choose the mode of excircle measurement

M03 ; Start the spindle of emery cutter

G04 X1 ; Delay 1 second

G00 X10 Z1 ; To the nearby face measurement

G31 U-3 F100 P1027 ; Excircle rough grinding in face enough M22, make sure within 3mm

G31 U-2 F10 P1026 ; Excircle fine grinding in face enough M24, make sure within 2mm

G31 U-1 F1 P1025 ; Excircle fine polish grinding in face enough M28, make sure within 1mm

G0 X10

M02

3、Compensate the abrasion of emery cutter

T0202

G00 X100 Z100 ; Must be the fixed point of emery cutter

G22 L5

GO U-0.01 ; Fix 0.05mm

G01 W-20 F10

W20

G800

G185 X100

M02

4. setting for high-speed punching machine, there are two high-speed input point, Other parameter P32=28, P910=1;

T0101

G00 X0

G22 L50 ; loop times

WAT+M295 ; punch up in position, detect input point M18(X28)

OUT-M295

G0 U1

WAT+M297 ; punch down in position, detect input point M14(X30)

OUT-M297

G800

M02

4.7 Instruction of following axis

1.Method A: the coordinate of follow-up axis will not follow, while the fourth axis can be hidden in screen. When axis parameter P414 is 7, the fourth axis is following X axis in both automatic and manual mode. When P414 is 8, the fourth axis is following Y axis in both automatic and manual mode. When P414 is 9, the fourth axis is following Z axis in both automatic and manual mode.

When P414 is 17/18/19, the C axis is following X/Y/Z axis in both automatic and manual mode. When P414 is 27/28/29, the B axis is following X/Y/Z axis in both automatic and manual mode.

Besides, D5...D13=1 means A axis follows X/Y/Z, C axis follows X/Y/Z, B axis follows X/Y/Z, so three axis can follow at the same time.

2. Method B: it is applied to specify follow-up axis with instructions in automatic mode, but not follow in manual mode. In this way, is coordinate following when axis is following.

M123: turn on the Y axis following X axis, Y axis is following with radius, so Y axis should programme with radius, please note.

M124: turn off the Y axis following X axis;

M125: turn on the A axis following X axis, A axis is following with radius, so A axis should programme with radius, please note. ;

M126: turn off the A axis following X axis.

Attention:

1> both finishing process turret and rough process turret set tool according to real measurement, finishing process turret is influence by User parameter “16,X program mode[1 mean Radius,0 mean Diamete]”, and rough process turret always works with radius program, pay attention when set tool.

2> finishing process turret X axis track will be defined by X coordinate in program. When add M123/M125 in program, rough proccessing turret will go forward a distance in Y axis or Axis direction, meanwhile rough process allowance will be left in the direction of work piece radius, which is defined by axis parameter P502.

3> when add M123/M125, please pay attention to not make finish or rough process turret touch work piece, because during execution of M123/M125, rough process turret will position at target point with G00 speed in Y axis or A axis direction according to current Z coordinate.

4> program running finish process turret primarily, and rough process turret will follow in Y or A axis, so the speed of rough process turret will wave, whose acceleration speed is defined by Y axis or A axis acceleration speed parameter. Low acceleration speed may influence accuracy of rough processing.

5> when rough process turret moves in negative direction of Y/A axis, it is feed tool, in order to prevent tool crash because of fast feeding speed, set axis paramtere P503 to limit speed. And too low value of P503 may influence accuracy of rough processing.

6> when rough process turret moves in positive direction of Y/A axis, it is retract tool, then the max speed is limited by G00 speed parameter.

7> rough process turret Y axis should A xis should be linear axis, instead of rotary axis.

8> it is better t enable smooth function to improve movement stability.

9> when Axis paramater P501 “M123/M125 following instruction Z axis forward distance” is effective, and DXF is set to do automatic sequence, then when translate DXF file into G coder file, controller will generate M123/M124/M125 automatically.

Program sample:

G0 X500 Z890 Y960 M3 S2000

M123 ; enable Y axis follows X axis

G1 X_ Z_ F_

X_ Z_

.....

.....

X_ Z_

M124 ; cancel Y axis following

G0 X400 Y800 ; move out from work piece.

M02



Ideal and Practical CNC Products Manufacturer



NEWKer Website



NEWKer Youtube

CHENGDU NEWKer CNC-TECHNOLOGY CO., LTD

Address: Economic Headquarter E2, No.888 Chenglong Avenue,
Longquanyi District, Chengdu City, Sichuan Province,
the People's Republic of China.

Website: WWW.NEWKER-CNC.COM

Email: NEWKer.CNC@aliyun.com

Phone: 028-84601148

Fax: 028-84609226

Skype: Newker-CNC James

Whatsapp: +8618990029555