

User Manual

NW-990TDb

Lathe Control System

CONTENTS

Chapter 1	Preface.....	1
Chapter 2	System technical features.....	2
2.1	System constructions.....	2
2.2	System technical parameter.....	2
2.3	System function.....	2
2.4	System operation condition.....	3
Chapter 3	Operation explanation.....	4
3.1	Panel layout and switch (Two types)	4
3.2	Keyboard description.....	5
3.3	Manual Operation.....	7
3.4	Auto operation.....	15
3.5	Operate safety, prompt alarm.....	19
3.6	Parameters.....	21
3.7	Set parameter of tool redeem.....	41
3.8	Screw compensation.....	46
3.9	Input/output diagnosis.....	50
3.10	Operation of program.....	52
Chapter4	Programming.....	58
4.1	basic concepts.....	58
4.2	Program instruction.....	59
4.3	Preparation functions.....	65
(1)	Rapid motion(G00).....	66
(2)	Line interpolation(G01).....	67
(3)	Arc interpolation(G02/G03).....	68
(4)	Screw thread (G32).....	69
(5)	Circularity screw thread(G332、 G333).....	72
(6)	delay Instruction(G04).....	72
(7)	Return Reference (G28/G281/G282/G283/G284/G285 /M800 / M881).....	73
(8)	Setup workpiece coordinate system (G50).....	74
(9)	Column or taper loop(G90).....	74
(10)	End face loop(G94).....	76
(11)	Screw thread loop (G92).....	79
(12)	Usage for fixed cycle.....	83
(13)	Fixed cycle of tapping (G93).....	85
(14)	Column thick loop (G71).....	86
(15)	End face thick loop(G72).....	88

(16) Close cutting loop(G73).....	89
(17) Finish machining loop (G70).....	90
(18) End face deep hole loop(G74).....	92
(19) Slot cutting loop(G75).....	93
(20) Complex screw thread loop(G76).....	94
(21) Fix loop(G22、 G800).....	98
(22) Part coordinate setup(G52), Setup tool coor (G184、 G185)	99
(23) Back start point(G26、 G261、 G262、 G263、 G264).....	101
(24) Back to G25 point(G25、 G61、 G611/G615).....	101
(25) continue feed cutting(G60/G64).....	101
(26) Constant speed cutting(G96/G97).....	101
(27) Feed mode(G98、 G99).....	103
(28) Check skip(G31、 G311).....	103
(29) Work coordinate(G53/G54/G55/G56/G57/G58/G59).....	103
(30) Tool radius compensate G40、 G41、 G42.....	104
(31) Radius compensation of tool C.....	106
(32) Automatical beveling (I) and smoothing(R).....	111
(33) Pole coordinate program(G15/G16).....	112
(34) Switch millimeter and inch(G20/G21).....	113
(35) Call sub-program (M97/M98/M99).....	113
(36) Conditional wait, jump instruction.....	115
(37) T tool.....	116
(38) S、 SS SP speed.....	116
(39) F feed speed.....	116
(40) Macro program instruction (G65、 G66、 G67)	117
(41) User-defined macro instruction(G120-G160,M880-M889).....	120
4.4 Synthetic instance for programming.....	122
4.5 Usage for grinder.....	123
Chapter 5 System installation and connection.....	125
5.1 system installation and connection.....	125
5.2 system installation dimension.....	125
5.3 system rear view.....	126
5.4 interface connection graph.....	126
Chapter 6 System's daily maintenance and repair.....	135
6.1 System's maintain.....	135
6.2 Ordinary trouble.....	135

Chapter 1 Preface

This CNC control system is one middle class flush type CNC control system, aiming specially at lathe and grinding machine.

Based on modern computer technology, system move control core & PLC program running technology, and stable unique real time control engine subsystem PTAI, this system ensures the stabilization of operation. The use of high performance, low power consumption industrial grade ARM microprocessor as core of hardware, large scale FPGA integrate circuit, multiple layer (4,6) printed circuit, 32MB flash memory, 8 inch real color LCD which provides friendly man-machine dialogue interface makes this system work to its best.

Note for “caution”:

1.“caution”reminds operator must be caution in the relative operation , otherwise the operation will fail or some action can not be effected.

2.“special caution”reminds operator must be special caution in the relative operation, otherwise it may break down the machine or give rise to accident.

Special hint:

This system has function to backup parameters. After debugging machine, it can backup all parameters of machine & system and PLC documents to computer. It is convenient not only for mass debugging, but also for machine recovery to normal after changing system.

Note :

When use this system for the first time, please read carefully all the details of each chapter so as to make it work more efficiently.

The functions which are related with 3rd ,4th,5th axis of system only exist in the controller that user order this functions.

3. The “Run” button on the panel of system can be used when debugging (No.9 parameter in other parameter to set “Effective” “Invalid”), must plus an external “Run” button when fitting system, otherwise may cause accident because of the life of button!!!So the system prohibits using the button for many times, otherwise the consequences has nothing to do with my company.

4.Must be de-energized when connect/plug with ports.

Chapter 2 System technical features

2.1 System constructions

32 bits high performance, low power consumption industrial grade ARM microprocessor.

64MB memory.

32Mb user store room.

640x480 8 inch real color LCD Displayer.

Touch screen main and sub panel.

High anti-jamming switch power.

USB movable U disc copy interface.

RS232 interface.

Spindle servo speed control/spindle frequency convention speed control.

Manual pulse generator.

2.2 System technical parameter

controllable axes: X、Z、C/Y、A four axes.

linkage axes: Arc 2-3 axes, liner 2-4 axes.

pulse equivalent: X、Z、C/Y、A axes:0.001mm.

max speed: X、Z、C/Y、A:60000mm/min.

cutting speed: 1-10000mm/min.

min input unit: 0.001mm.

program size range: ± 99999.999 .

99 tools management.

controllable liner vertical type or revolving disc type tool changer.

program code: ISO-840 international standard.

program coordinate system definition: ISO-841.

chassis protection complies with regulation of IP43.

2.3 System function

2.3.1 Auto-diagnosis function

All around diagnosis of CPU, storer, LCD, I/O interface, parameter status, coordinates, machining program etc. shall execute when the system starts or resets. In operation, it makes real time diagnosis of power, spindle, limit and all I/O interface.

2.3.2 Compensation function

automatic backlash compensation.

tool radius automatic compensation.
tool radius automatic offset and sharp angle transition.
leading screw pitch error automatic compensation.

2.3.3 Abundant instruction system

scaling up/down instruction.
mirror machining instruction.
Multiple tool offset instruction.
program cycle, jump, call and different program ending.
multiple positioning instruction: starting point, setting fixed point, etc.
Linear, circular, spiral line interpolation instruction.
program management instructions: program cycle, call, transfer and different program ending method, etc.

6 workpiece coordinates system .

2.3.4 Chinese/English menu, full screen edition

Easy operation, convenient viewing.

2.3.5 Abundant debugging functions

it can point out clearly what errors of operation are and guide to correct them.

2.3.6 Program changing between CNC system and IBM/PC series compatible computer

it can conduct CAD/CAM/CAPP auxiliary programming by using Pc series compatible computer's abundant software resources , then transfer the CNC program into the system to machining through (USB movable U disc copy port、RS232 port) .Likewise it also can transfer the program from system to PC through communication port.

2.4 System operation condition

2.4.1 Power supplying

AC 220V(+10%/-15%), Frequency 50Hz±2%. power: ≤ 200W.

Note: it must use isolation transformer to supply power, first input:380V

2.4.2 Climate condition

Operation condition: temperature 0~45℃, relative moisture 40-80%.

storage & transportation condition: temperature -40~55℃, relative moisture <93%(40℃).

atmosphere pressure: 86-106kpa.

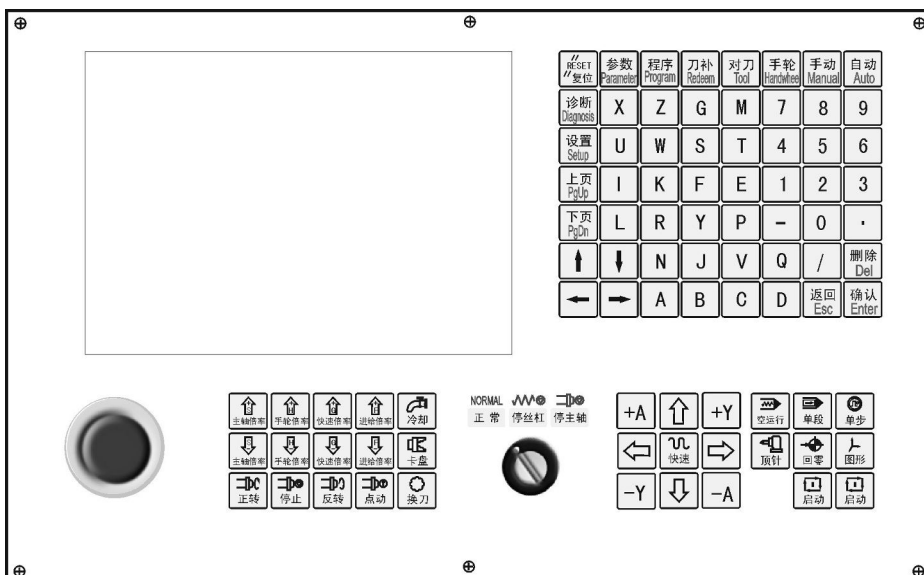
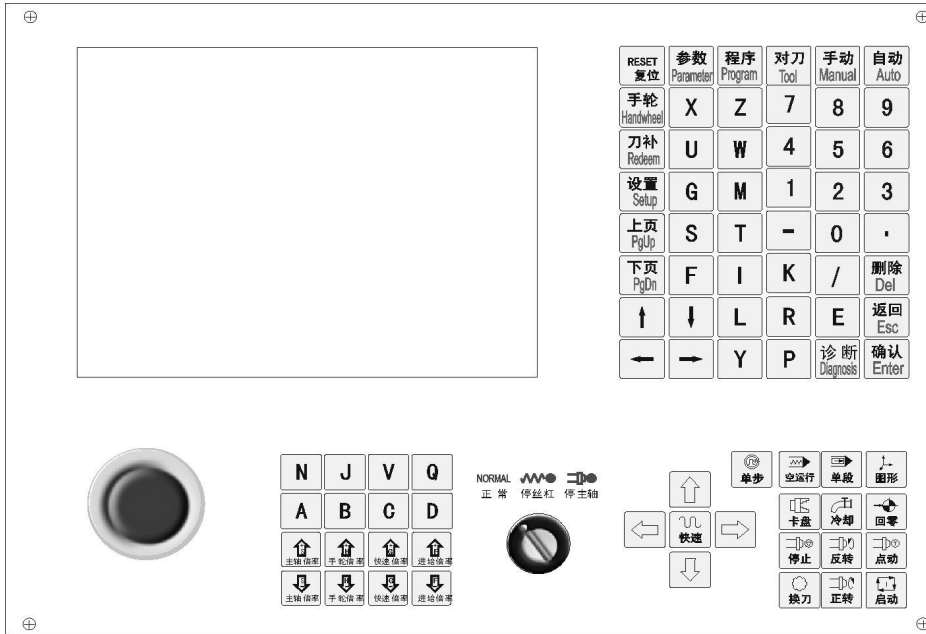
2.4.3 operation environment:

No excessive flour dust, no acid, no alkali gas and explosive gas, no strong electromagnetic interference.

Chapter 3 Operation explanation

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

3.1 Panel layout and switch (Two types)



3.2 Keyboard description

3.2.1 Rate increase or decrease

(1) Rapid override(G)

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

(2) Feed override(F)

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

(3) Spindle override(S)

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

3.2.2 Usage for intervention switch

switch introduction:

Switch	Functions
	Emergency stop Driver and motor stop immediately, turns off the spindle, coolant, waits for the rise of button, and initializes values
 	(1) The left: normal processing. (2) The middle: manual to stop feeding, suspend the automatic feeding. Return to the left to the normal status. (3) The right: suspend automatic feeding. Return to the left to the normal status. Spindle is out of control when running with non-coordinate axis.

Buttons introduction:

Keyboard s	Functions
Letter key Number key	ABCSEFGHIJKLMNOPQRSTUVWXYZ123456789.-: for program instructions, parameters' edition; 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	“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 speedy simulating  ”for manual increment or constant work
Control key	    ”spindle cw, ccw rotation  ” coolant on/off  ”for the shift between electric tool carrier and gang tool carrier  ” for the shift between hand-driven continuous high speed and low speed.  ” all axes return to datum point  ” for spindle chuck on/off  ” for thumbstall on/off   ”handwheel gear selection   ” adjusting spindle speed   ”adjusting feed speed   ”adjusting G00 speed
Feed key	    +Y +A-Y -A For X、 Y、 Z 、 A axes direction feed

3.3 Manual Operation

The system adjusts one-level menu operation, intuitive, convenient, shortcut, prompt comprehensive information. Powering the system is to enter the main interface:

Man Con		N00000		2012-12-21 19:26	
N1 M03 S600			Program 99.TXT		
N2 G00 X200 Z200			Instruction code		
N3 G00 U-44			G53		
			T0001		
			Machine Status		
			M05 M09 M10		
			M78 M33 M70		
			G00 X100%		
			F6000 X100%		
			S50 X100%		
			Machine Coord		
			X 166.000		
			Z 161.096		
			C 0.000		
			PartTime 0: 15		
			PartNo 17		
			SPrpm 0		
No Alarm					

In the screen, it is run mode in left of first line (run mode: “Manually continuous”, “Manually increment”, “automatic continuous”, “automatic single-stage”, “automatic air operation”, “handwheel”), line number of processing in middle, time is in right.

It is program area in upper left, second line is processing program line in program area.

It is coordinate area in bottom left. Penultimate line is prompt area, such as alarm note. In last line, it is display the “F1-F8” function key.

They are current program name, code, machine condition, machine coordinate, etc in the right.

Press “Program” key enter program management area. It could edit, alter, diagnosis, delete, and copy etc.

Press “Parameter” key enter parameter management area. It could refer, alter to every parameters.

In manual condition, the system could process workpiece.

3.3.1 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 30000 mm/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 65000, 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 handwheel 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.

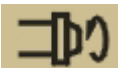
(15) “Chuck”: Chuck tightens or loose correspond to instructions M10,M11.

(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) “Tailstock”: Tailstock tighten or loose correspond to instructions M79, M78.

1) manual main axle condition:

Press  the principal axis veer,display M03.

Press  the principal axis reverse, display M04.

Press  the principal axis stop, display M05. At this condition, users can press the key to turn off or turn on.

Press  M03 turn on point for while.

Press  the coolant to turn on or turn off.

Press  SP chuck to turn on or turn off.

Press  in choosing the position cutting tool.

Press  thumb stall to turn on or turn off.

2) Adjust the feed speed: The feed speed percentage can be controlled by the wave

band switch or the key , the percentage increases or decreases 10%. The scope is 0 -150%, 16 grades in all.

3) Control the principal axis speed: The main axle speed percentage can be

controlled by the wave band switch or the key , the percentage increases or decreases 10%. The scope is 0 -150%, 16 grades in all.

4) presses "stops": Stops the manual operation.

5) presses "F", there's a dialog box used to alter the manual feed speed. That is convenient for cutting by single axis.

6) presses "S", alter the principle axis's revolving speed.

7) presses "T", choosing the position cutting tool.

(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 datum point”: Press  and X or Z, the X or Z axis goes back to the datum point automatically; Press “0” X axis firstly and then Z axis; Press “Esc” to cancel the 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) “Partno clear”: Press Del and Enter.

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

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

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

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

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

3.3.2 Manual continuous

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

When continuous starting, press “” to switch 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.3 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.

3.3.3 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.

The speed is the speed on display(F) times the rate.

3.3.4 Back to lathe's datum point (reference point/Home/zero point)

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

Switch for datum point:

Back to datum point operation is to feed every axis to lathe's datum point position in turn. When the parameter of feeding axis which back to datum point is 0, the axis of coordinate detects the datum 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 datum point to set lathe's coordinate correctly.

Instruction:

1. The system requires for backing to the datum point every time when it is power on, the requirement can be set by No.26 parameter in axis parameter, it can be prompt or force;
2. The way and type of detecting signal can be set by No.27 parameter in axis parameter, so detect the switch of datum point is effective, also detect the Z pulsing signal of electrical motor after detecting the switch of datum point (precision higher), detect forward or reverse for Z pulsing signal of electrical motor.

3. The direction for backing to datum point can be set by No.28 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 datum point can be set by No.28 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 datum point can be set by No.29 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.30 No.31 parameter in axis parameter, the value must less than the pulse of electrical motor run a cycle.
7. The shifting distance after backing to datum point can be set by No.32 No.33 parameter in axis parameter, rapid move coordinate to the value of parameter after backing to datum point.

No switch for datum point: To set floating point to make sure, turn on corresponding function of floating point by No.23 parameter in axis parameter, setting No.24 No.25 parameter to make sure X axis' and Z axis' floating point, the datum point of lathe.

The steps to set floating point as follows:

1. Setting the No.23 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.24 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 datum point of X axis. The lathe backs to this position every time when backing to the datum point.

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

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

Operation for backing to the datum point:

At the manual condition, press "-return" and select X Y Z A B axis to back to the datum point in dialog box. Set the No.28 parameter in axis parameter to "1" to make Z axis back to the datum point first. At the absolute and relative coordinate condition, the cycle will turn to green in front when backing to the datum point successfully, defeat otherwise.

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

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

3.3.5 Handwheel(Manual pulse generator)

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

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 pulse generator speed to be lower than 200r/min(The handwheel to 100 pulse a cycle), the Handwheel acceleration is controlled by No.17 parameter in speed parameter(the bigger the faster). The maximum speed is controlled by No.20(X axis) No.21(Z axis).

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

3.3.6 Alignment Tool

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:

Method A (recommend)

- (1) Make sure the “Tool” axis couldn’t exit.
- (2) The tool must touch workpiece.
- (3) Use “Z” axis’ direction to “Tool”

Method 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 “F8” 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 “F7”, “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 “F7”, “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 okay.

The method: In the G53 status, switch “F6” into “F7” as the above operations.

3.4 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 “F7” to select the program to carry out automatically.

Switch the display of coordinate: Press “F1-F3” to switch the display which correspond to “relative” “absolute” “machine”.

3.4.1 Automatical 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.

3.4.2 Processing at arbitrary program line or with arbitrary tool

A. Import the line to run

At the automatical process condition, press “—” to popup a dialog box, import a number of line, 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.5 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. Mark the line to run

The system has a function to run at the marking line. At automatical process condition, press “N” to popup 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.5 parameter in speed parameter(G01/G02/G03), then process the program normally.

C. Some tool to run

The system has a function to run at some tool. At automatical 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.5 parameter in speed parameter(G01/G02/G03), then process the program normally.

3.4.3 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.

3.4.4 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)

3.4.5 Real-time control in automatic process

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

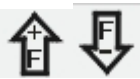
Left: Feeding coordinate and spindle is not limited.

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 automatical 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.

3.4.6 The operation mode of MDI

At the manual or automatical coordinate conditions, press “MDI” 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.

3.4.7 The operation mode of Handwheel

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

Pay attention: The acceleration, deceleration and maximum speed of running handwheel are controlled by No.17 No.18 No.19 No.20 No.21 No.43 parameter in speed parameter and No.23 parameter in processing parameter, use the acquiescent acceleration, deceleration and the speed of G00 when the parameter is set to be invalid.

3.4.8 The function of DNC

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 for 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 “F6” to open U-disk in program interface, select corresponding program to press “F7” 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.

3.5 Operate safety, prompt alarm

3.5.1 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 datum 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.

3.5.2 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 datum point.

3.5.3 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 forcedly : X or Z axis is in the positive position which is limited forcedly.

(2) X and Z axis are limited negative forcedly: X or Z axis is in the negative position which is limited forcedly.

(3) Spindle and inverter (frequency changer) alarm: The alarm signal of lathe’s inverter is effective. (ALM1)

(4) No.0 alarm: The alarm signal of lathe’s spindle 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.

3.6 Parameters

At any status conditions, press “parameter” to enter the status to set the parameter. Parameter in this system includes “processing parameter” “speed parameter” “axis parameter” “tool parameter” “other parameter” “coordinate” “password”, 7 kinds totally.

Man Con		N00000		2012-12-21 19:26	
1,Cycle G71/G72		50		Program 99.TXT	
2,Cycle G71/G72		10		Instruction code	
3,G71G72G173 i		0		G53	
4,G71/G72/G73 d		20		T0001	
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		F6000 X100%	
10,G74/G75 defa		10		S50 X100%	
11,G76 finish t		1		Machine Coor	
12,G76 quit len		10		X 166.000	
13,G76 thread t		60		Z 161.096	
				C 0.000	
N-User	J-Speed	V-Axis	Q-Tool	PartTime	0: 15
A-Other	B-Coor	C-PASSWD	D-CANCEL	PartNo	17
No Alarm				SPrpm	0

In main menu, pressing“Parameter”function key, it enters para setting status,including“User”,“Speed”,“Axis”,“Tool” ,“Other” ,“Coor>>” ,“Passwd”,seven function. Choose pressing “N、J、V、Q、A、B、C” choose

Except for special note, all data are using mm.

3.6.1 User parameter

- 1,Cycle G71/G72 default feed thickness(10um) [X axis radius]
Cycle G71 X axis feed thickness; Cycle G72 Z axis feed thickness;
- 2,Cycle G71/G72 default backward distance(10um) [X axis radius]
Cycle G71 X axis backward thickness; Cycle G72 Z axis backward thickness.
- 3,G71G72G173 instruction [1 mean Yes,0 mean No]
“1”mean G71/G72/G73 instruction finish machining.
- 4,G71/G72/G73 default X remain(10um)
- 5,G71/G72/G73 default Z remain(10um)
- 6,G73 cutting times
- 7,G73 X rough thickness(10um) [X axis radius]
- 8,G73 Z rough thickness(10um)
- 9,G74/G75 default feed thickness(10um) [X axis radius]

- 10,G74/G75 default backward distance(10um) [X axis radius]
- 11,G76 finish turn times
- 12,G76 quit length(1/10 lead)
- 13,G76 thread tooth angle(degree) [0~180°]
- 14,G76 minimal cutting depth(10um) [X axis radius]
- 15,G76 finish turn remaining(10um)
- 16,X program mode [1 mean Radius,0 mean Diameter]
- 17,Running program need Sp run [1 mean Yes,0 mean No]
- 18,Set M20 the time of auto-running [Negative number mean immensity loop]
- 19,Set part count
- 20,G92 quit length(1/10 lead)
- 21,G01/G02/G03 line delay(ms)[>100]
- 22,G00 line delay(ms)[>100]
- 23,Handwheel acceleration[50-100]
- 24,G76 Q [8 mean thick forward number]
- 200,system screen protect times [>=2minutes]
- 201,G92/G76 delay time(ms)[>100]
- 202,system inner parameter

Explanation of User Parameter(processing parameter)

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

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

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

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

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

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

3,G71G72G173 instruction

It is for whether smoothing automatically when use G71/G72/G73 code.

When set to 0,mean no; when set to 1,mean yes.

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

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

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

It is set for every default remain of smoothing(diameter) in Z-axis direction

when it isn't be set in G71/G72/G73 code;unit:0.01mm.

6,G73 cutting times

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

7,G73 X rough thickness(10um)

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

8,G73 Z rough thickness(10um)

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

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

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

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

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

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

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

11,G76 finish turn times

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

12,G76 quit length(1/10 lead)

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

13,G76 thread tooth angle(degree)

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

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

It is for set minimal cutting depth (radius) of G76.Unit:0.1mm

15,G76 finish turn remaining(10um)

It is for default remaining of finish turn when it isn't be set in G76 code.unit:0.01mm.

16,X program mode [1 mean Radius,0 mean Diameter]

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

17,Running program need Sp run [1 mean Yes,0 mean No]

It is for interlock between run program and run spindle,when set as 1 means that running program with running spindle;when set as 0 means that running program without check spindle running.

18,Set M20 the times of auto-running [Negative number mean immensity loop]

It is for times of run M20 code in the program,negative number mean run

countless times.

19,Set part count

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

20,G92 quit length(1/10 lead)

It is for set default length of quit and retract,the length=thread lead * 0.1.

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

It is for set delay time between G01/G02/G03,it is for solve the over-cutting in the corner.

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

It is for set delay time after run G00 ,it is effective that more than 100ms.

23,Handwheel acceleration[50-100]

It is for set the constant of handwheel smoothly acc/dec-elation. the smaller it is,the faster the acc/dec-elation is,but much vibration.

24,G76 Q [8 mean thick forward number]

It is for set the define of Q in G76,set as “8”,it is the times of feeding in roughing.

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

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

It is for set delay time before check Z pulse when process screw.

3.6.2 Speed parameter

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

2,Z-axis's G00 speed(mm/min)

3,Manual maximum feed speed(mm/min)

4,Auto Maximum feed speed(mm/min)

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

6,Null run speed(mm/min)

7,Feed axis`s manual speed(mm/min)

8,Spindle`s manual speed(rpm)

9,Beginning feed speed(mm/min)

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

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

12,X G1G2G3 max speed(mm/min)

13,Z G1G2G3 max speed(mm/min)

14,X acceleration [1~99999]

15,Z acceleration [1~99999]

16,Auto run acceleration [1-500]

17,Handwheel acceleration [500--30000]

18,Run program Handwheel acceleration [>500]

- 19,Run program Handwheel G00 speed(mm/min) [>10]
- 20,Handwheel X limit speed(mm/min)
- 21,Handwheel Z limit speed(mm/min)
- 22,Make thread Z acceleration [Servo motor is 0]
- 23,Make thread X acceleration [Servo motor is 0]
- 24,Servo motor screw thread X axis Back speed
- 25,Step motor screw thread X axis Back rise speed
- 26,Step motor screw thread X axis Back Max speed
- 27,acceleration type [0 mean line,8 mean curve]
- 28,curve ini acceleration [≥ 10]
- 29,curve acceleration [≥ 10]
- 30,curve max acceleration [≥ 500]
- 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)
- 35,G96 spindle min speed(rpm)
- 36,Spindle first max speed(rpm)
- 37,Spindle second max speed(rpm)
- 38,Spindle third max speed(rpm)
- 39,Spindle forth max speed(rpm)
- 40,Second Spindle max speed(rpm)
- 41,G02/G03reverse compensation mode(0 mean A; 8 mean B)
- 42,mode B reverse compensation speed(mm/min)
- 42-1,mode B reverse compensation Beginning feed speed(mm/min)
- 42-2,mode B reverse compensation acceleration(mm/min/s)
- 43,Handwheel stop speed(mm/min)[>100]
- 58,Forcedly limit drop speed critical(mm/min)

Introduction:

- 1,X-axis's G00 speed(mm/min) 2,Z-axis's G00 speed(mm/min)

The max is 30000(unit:mm/min)

Note:the value must take machine configuration into consideration ,set wrong is very dangerous.

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

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

- 4,Auto Maximum feed speed(mm/min)

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

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

It is the default speed of G01/G02/G03 when it isn't set in auto-running.Max:5000(unit:mm/min)

6,Null run speed(mm/min)

It is speed of null run. (press “simulate” is dry run) Max:30000. (unit:mm/min)

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

It is the speed of feeding axis in Manual.Range: <max feeding speed (unit:mm/min)

Note:in Manual,press “F” ,refresh the parameters automatically.

8,Spindle's manual speed(rpm)

It is speed of spindle in manual.Unit:rpm.

Note:in Manual,press “S” ,refresh the parameters automatically.

9,Beginning feed speed(mm/min)

It is beginning speed of feeding axis when acc/dec-elation.when it is smaller than acceleration/ deceleration ,accelerate/decelerate start from the beginning feed speed.when it is bigger than acceleration/deceleration,the speed reached directly.

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

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

It is for increase the continuous when running multiaxial track-interpolation.

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

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

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

12,X G1G2G3 max speed(mm/min)

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

13,Z G1G2G3 max speed(mm/min)

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

14,X acceleration [1~99999]

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

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

15,Z acceleration [1~99999]

It is time constant of Z-axis acc/dec-elation,the bigger it is ,the faster the

ace/dec-elation is.

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

16, Auto run acceleration [1-500]

It is for set constant of acc/dec-elation in auto. the range is from 1-500. It is mainly for distinguish Auto and Manual, only the difference is too much, set it is effective.

17, Handwheel acceleration [500--32000]

It is for set constant of acc/dec-elation of Handwheel. the range is from 500-32000.

18, Run program Handwheel acceleration [>500]

It is for set constant of acc/dec-elation of Handwheel when running program. the range is from 500-32000. when the value is less than 500, it is invalid.

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

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

20, Handwheel X limit speed (mm/min)

It is for limit the handwheel max speed of X-axis when use handwheel in manual.

Note: it is valid when >100 , otherwise invalid.

21, Handwheel Z limit speed (mm/min)

It is for limit the handwheel max speed of Z-axis when use handwheel in manual.

Note: it is valid when >100 , otherwise invalid.

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

It is time constant of Z-axis ace/dec-elation, the range is from 300 to 90000.

Note: with stepper system, the smaller it is, the safer is. when <300 , invalid. For ensure efficiency, set 0.

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

It is time constant of Z-axis ace/dec-elation, the range is from 300 to 90000.

Note: with stepper system, the smaller, the safer. when <300 , invalid. For ensure efficiency, set 0.

24, Servo motor screw thread X axis Back speed

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

25, Step motor screw thread X axis Back rise speed

It is the start speed of X axis in step system when back in processing screw thread. Unit: mm/min.

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

26, Step motor screw thread X axis Back Max speed

It is the Max speed of X axis in step system when back in processing screw thread. Unit: mm/min.

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

27, acceleration type [0 mean line, 8 mean curve]

It is for set type of acc/dec-elation. set 0 means line type. set 8 means curve type.

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

28, curve ini acceleration [≥ 10]

It is initial acc/dec-elation constant when set curve type. It is valid when ≥ 10 .

29, curve acceleration [≥ 10]

It is second acc/dec-elation constant when set curve type. It is valid when ≥ 10 .

30, curve max acceleration [≥ 500]

It is Max acc/dec-elation constant when set curve type.

It is valid when ≥ 500 , otherwise the acc/dec-elation constant is with line type of each axis.

31, X go home ramp speed (mm/min)

It is speed of X-axis when go home in forward direction. Unit: mm/min. the range is less than the G00 speed of X-axis.

32, X go home reverse speed (mm/min)

It is speed of X-axis when go home in reverse direction. Unit: mm/min. the range is 20-500.

Note: it is for ensure accuracy. the smaller it is, the higher the accuracy is. when set well, don't change it forever.

33, Z go home ramp speed (mm/min)

It is speed of Z-axis when go home in forward direction. Unit: mm/min. the range is less than the G00 speed of Z-axis.

34, Z go home reverse speed (mm/min)

It is speed of Z-axis when go home in reverse direction. Unit: mm/min. the range is 20-500.

Note: it is for ensure accuracy. the smaller it is, the higher the accuracy is. when set well, don't change it forever.

35, G96 spindle min speed (rpm)

It is the min speed of spindle when run G96 code.

36, Spindle first max speed (rpm)

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

37, Spindle second max speed (rpm)

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

38, Spindle third max speed (rpm)

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

39, Spindle fourth max speed (rpm)

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

40, Second Spindle max speed (rpm)

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

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

It is reverse compensation mode of arc gap.

0 means A mode. (A mode is the bigger it is, the faster the speed is. the speed should not be bigger than 1000 mm/min, the speed also is related with the value of reverse gap compensation.)

8 means B mode. (B mode is the speed depends on the related parameters.)

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

It is the speed of reverse compensation in B mode. unit: mm/min.

42-1, mode B reverse compensation Beginning feed speed (mm/min)

It is beginning speed of reverse compensation in B mode. it is valid when it > 10. unit: mm/min.

42-2, mode B reverse compensation acceleration (mm/min/s)

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

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

It is the speed when handwheel stop. the bigger it is, the shorter the stop time is.

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

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

3.6.3 Axis parameter

1, Feed axis band switch [1 mean Yes, 0 mean No]

2, Spindle band switch [1 mean Yes, 0 mean No]

3, X-axis's negative scope (mm)

4, X-axis's positive scope (mm)

5, Z-axis's negative scope (mm)

6, Z-axis's positive scope (mm)

7, Spindle stop time (10ms)

8, Spindle stop long signal [0 mean No, 1 mean Yes]

- 9, Check SP encode : [1 mean Yes, 0 mean No]
- 10, SP encode pulse
- 11, Soft limit invalid [D2X;D3Y;D4Z;D5A;0 invalid, 0 valid]
- 12, X-axis's reverse compensation(um)
- 13, Z-axis's reverse compensation(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]
- 16, Close feed electron gear [1 mean Yes, 0 mean No]
- 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)
- 21, XZ positive limit [0 open, 1 close]
- 22, XZ negative limit [0 open, 1 close]
- 23, float zero bit parameter [D3X;D4Y;D5Z;D6A;0 machine Zero; 1 float Zero]
- 24, X coordinate float zero set
- 25, Z coordinate float zero set
- 26, Feed axis home [1 mean No use, 0 mean clew, 8 compulsion, 9 must compulsion]
- 27, Feed axis home mode [0 reverse check, 1 reverse No check, 2 No reverse check, 3 No reverse No check]
- 28, Home reverse direction [D2X;D3C(Y);D4Z;D5A;D6B;D8Zpriority]
- 29, Home NC switch bit set [D0X;D1C(Y);D2Z;D3A;D4B;D7:Manual/Auto cut automatically ; 1Close; 0Open]
- 30, X check zero max length(100um)
- 31, Z check zero max length(100um)
- 32, X Home offset(10um)
- 33, Z Home offset(10um)
- 50, Have Spindle class control : [1 mean open, 0 mean close]
- 51, Spindle class speed(1/100rpm)
- 52, Spindle class direction : [0 mean M03, 1 mean M04]
- 53, Spindle class stop time(10ms)
- 54, Spindle class time(10ms)
- 55, Spindle stop time(10ms)
- 80, XZ axis coordinate plan[D2Zwordpiece, D3Xwordpiece, D4Ztool, D5Xtool, D6Zcircumrotate, D7Xcircumrotate]
- 81, Y axis [0 mean circumrotate axis, 1 mean line axis]
- 82, Y is circumrotate axis work coordinate : [0 No; 1 plan]
- 83, Y is circumrotate axis machine coordinate : [0 No; 1 plan]
- 101, 3rd-axis name[0:Y, 1:C]
- 102, lathe C axis function[0 mean circumrotate axis, 1 mean linear axis]
- 103, lathe C is circumrotate axis, the machine coordinate : [0 Null; 1 absolute coordinate plan, 2 tool coordinate, 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) backlash compensation(um)
 108.C(Y)-axis's G00 Speed(mm/min)
 109.C(Y)-axis's 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 length(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) coordinate float zero set
 200.inner parameter
 404,SP motor direction(0 reverse,1 normal)
 405,SP-axis's electron gear(0 Yes,1 No)
 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,Interpolation tap SP name[91 X,92 Y/C,93 Z,94 A,95 B]
 411,Interpolation tap mode[2 follow encode;3 interpolation to SP]
 412,SP tooth number(<P413)
 413,Encode number(>P412)

Introduction:

1,Feed axis band switch [1 mean Yes,0 mean No]

It is for operation way of alter feeding axis's rate.1 mean selection external band switch(it is the band switch in additional panel.) to alter.0 mean use Feed rate+/-.

2,Spindle band switch [1 mean Yes,0 mean No]

It is for operation way of alter spindle axis's rate.1 mean selection external band switch(it is the band switch in additional panel.) to alter.0 mean use SP rate+/-.

3,X-axis's negative scope(mm)

It is the coordinate value of X-axis soft limit in max scope of negative direction.

4,X-axis's positive scope(mm)

It is the coordinate value of X-axis soft limit in max scope of positive direction.

5,Z-axis's negative scope(mm)

It is the coordinate value of Z-axis soft limit in max scope of negative direction.

6,Z-axis's positive scope(mm)

It is the coordinate value of Z-axis soft limit in max scope of positive direction.

7,Spindle stop time(10ms)

It is the braking time of spindle,the shorter it is,the faster the brake is.

8,Spindle stop long signal [0 mean No,1 mean Yes]

When it is 1,the signal of spindle is long signal,when it is 0,it is short signal.

9,Check SP encoder : [1 mean Yes,0 mean No]

It is for whether the system check the signal of spindle encoder,also the spindle's position feedback.

When 1 mean check ,0 mean no check.the spindle gear ration must be 1:1 with spindle encoder.

10,SP encode pulse

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

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

It is invalid bit parameter of soft limit.every axis is set alone.D2:X;D3:Y(C);D4:Z,D5:A;D6:B.

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

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

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

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

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

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

It is for change the direction of X-axis.when it is 0,the direction of code is opposite to the direction of moving.when it is 1,the direction is same.

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

It is for change the direction of X-axis.when it is 0,the direction of code is opposite to the direction of moving.when it is 1,the direction is same.

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

It is for whether close the electron gear of feeding axis.1 mean close,0 mean no close.

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

It is the numerator of X-axis's electron gear,Multiplication ratio of axis X's instruction(X_CMV)

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

It is the denominator of X-axis's electron gear ,Frequency-division coefficient of axis X's instruction(X_CMD)

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

It is the numerator of Z-axis's electron gear,Multiplication ratio of axis Z's instruction(Z_CMV)

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

It is the denominator of Z-axis's electron gear,Frequency-division coefficient of axis Z's instruction(Z_CMD)

P17-P20 parameters:

Effective Range: 1-999999

Unit:non

User:Upon operating administrators

Initialization:1

Effective time:Immediately

Explain:

When lead screws with different screw pitches are configured with motors of

various step angles, or with servo motors of different pulse number per round, or connections are realized through different gears, the programmed values can remain consistent with the actual moved distance by setting the parameter of the electronic gear ratio of the system.

$$\text{CMR}/\text{CMD} = P / (L * 1000)$$

CMR: Numerator of gear ratio

CMD: Denominator of gear ratio

P: pulse number per motor round

L: Moved distance per motor round(mm)

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

Example1:

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

$$\text{CMR}/\text{CMD} = 5000 / (5 * 1000) = 1 / 1$$

That is to say, we can set the values as :CMR=1, CMD=1.

Here, the pulse equivalent is 0.001mm.

Example2:

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

$$\text{CMR}/\text{CMD} = 5000 / (10 * 1000) = 1 / 2$$

That is to say, we can set the values as :CMR=1, CMD=2.

Here, the pulse equivalent is 0.002mm.

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

It is for type of limit switch in positive direction. 0 means the switch is open, 1 means it is close.

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

It is for type of limit switch in negative direction. 0 means the switch is open, 1 means it is close.

23, float zero bit parameter [D3:X; D4:(C)Y; D5:Z; D6:A; 0 machine Zero; 1 float Zero]

It is for whether the float zero function is valid, every axis is set alone. It is bit parameter. D3:X; D4:C(Y); D5:Z; D6:A; D7:B. 1 means the axis is float zero point, 0 means machine zero point.

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

24, X coordinate float zero set

It is the coordinate value of X-axis float zero point.

25, Z coordinate float zero set

It is the coordinate value of Z-axis float zero point.

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

It is request that feeding axis go home. There are four kinds of go home as follow:

1 means no request. When boot every time, no prompt and no limitation;

0 means prompt, when boot every time, there will be a prompted screen;

8 means enforcement, when boot every time, there will be a prompted screen, and then, if the system don't go home, it will note "feed axis don't go home", and don't run

the program;

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

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

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

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

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

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

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

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

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

Example:when set X-axis go home in negative direction,the bit parameter is 100000100.

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

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

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

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

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

Note:the value must be less than the length of one rev,otherwise,go wrong home.

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

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

Note:the value must be less than the length of one rev,otherwise,go wrong home.

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

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

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

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

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

It is whether the spindle is booting when change gears.1 mean spindle is open,0

mean close.

51,Spindle class speed(1/100rpm)

It is the speed that the spindle boot when the spindle change gears.

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

It is the direction that the spindle boot when SP change gears,1 means reverse,0 means forward.

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

It is the time that the spindle stop (M05) when SP change gears.

54,Spindle class time(unit:10ms)

It is the time that the spindle run in low class.unit:10ms

55,Spindle stop time(unit:10ms)

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

80,XZ axis coordinate plan[D2Zworkpiece,D3Xworkpiece,D4Ztool,D5Xtool,D6 Z circumrotate ,D7Xcircumrotate]

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

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

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

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

It is for whether the spindle use electron gear.

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

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

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

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

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

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

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

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

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

It is the axis that be use for spindle when interpolation tap.

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

It is control mode of interpolation tap.

412,SP tooth number(<P413)

It is tooth number of spindle.it <=P413.

413,Encode number(>P412)

It is tooth number of SP-encoder,it >=P412.

Note:the tooth number of spindle must be not more than the tooth number of SP-encoder,when less,it need to install our company's adapter plate.

3.6.4 Tool parameter

1,Active tool function : [1 mean Yes,0 mean No]

It is for whether activate electric turret(tool).

Note:when the machine is with linear turret without electric turret,the parameter be set as 0,and also use Txxxx code to control tools and radius/tool compensation.

2,Active tool number of electric turret.

It is the total number of electric turret.

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

3,Lather type

It is type of lath machine’s structure.0:turret in front of horizontal lathe; 1:turret behind of horizontal lathe;2:turret in front of vertical lathe;3:turret behind of vertical lathe.

4,Tool positive rotate max-time(s)

It is the max-time that the turret is changing tool automatically.when over the time,the system stop ATC and alarm.

5,Delay time after tool positive rotate(ms)

It is delay time that check the tool signal(Tok) after the turret rotated

6,Delay time after tool stop(ms)

It is the delay time that is between the turret forward rotate is okay and stop,also is between cancel forward rotate signal(+T) and output reverse rotate signal(-T).unit:ms.

7,Tighten time of tool reverse rotate(ms)

It is the time tighten tool in reverse rotate, also is the time that output the signal of “-T”.

Note:the bigger the value is,the hotter the motor is.

9,Have total signal TOK(1 mean have)

It is for whether check the signal of “TOK”,1 means check it,0 means not.

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

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

20,Active tool mode :[1 mean normal,0 mean coding tool]

When set as coding tool,must restore in PLC of coding tool,detailed code and edit ATC program are press “F1”in Diagnosis condition.it needs to input password.

3.6.5 Other parameter

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

It is the type of handwheel,1 means hand held type,0 means handwheel in panel(C panel).

Note:when the parameter is 1(P1=1),CN11(handwheel port) couldn’t be used to alter axis,so P1,P2 only set as 0.A/X/Y(C)/Z to select axis,X1/X10/X100/OFF to

select grade.

2,lather outside chuck : [1 extroversion,0 diffidence]

3,use control switch : [1 Yes, 0 No]

It is for whether the tamper switch is valid.1 means valid,0 mean invalid.

4,Have auto lubricate(0 Yes/ 1 No)

It is for whether the auto lubricate is valid.1 means valid,0 mean invalid.

Note:auto lubricate is decided by working time.

5,Auto lubricate time(0.01s)

It is the time of auto lubricate,also time that M32 is valid.unit:0.01s.

6,Auto lubricate stop time(s)

It is the interval that lubricate every time,also the interval that twice M32 is valid.unit:s.

7,Door switch checking M12(0 no,1 yes)

It is for whether the system check the signal of safe-door.0 means there isn't safe-door,1 mean there is safe-door and check it.

Note:for the check of safe-door,it is realized by M12.

When set valid and M12 is also valid,in Manual,system could work,but in Auto,it not.

8,Door switch(Model :0 open,1 close)

It is type of safe-door.0 means normal open type,1 means NC(normal close).

9,bit parameter

D0: Null;

D1: "1"Start CNC system clear part Number.;

D2: "1"Automatic space before letter when edit program;

D3: Null;

D4: Null;

D5: "1"Do not stopping SP and cooling when pressing "Restet";

D6: "1"G00 XZ'speed by oneself;

D7: "1"Tool'redeem by oneself;

D8: "1"Save SP chuck(M10/M11) state when power off;

D9: Tool redeem input Mode1 or Mode2;

D10: "1" Program edit automatic compositor Line;

D11: "1"First SP +10V output from second output port;

D12: "1"Shield skip function ("/"is invalidation);

D13: "1"Shield go home function;

D14: "1"Shield "run" key;

D15: "1" Tool redeem display relative,"0"absolute;

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

It is for whether count workpiece automatically,1 means count,0 means not.

11,Program edit number increase

12,Inner parameter

13,Does lock for Spindle & chuck(0 mean no)

It is for whether interlock between running spindle and chuck.1 means interlock,0 means no.

14,Is available keys of lubricate&coolant as running(0 mean no)

It is for whether the coolant is valid in Auto.when P14=0,it is invalid.1 means valid.

15,Chuck clamp M10/loose M11 checking(1 mean need)

16,Finial forward M79/backward M78 checking(1 mean need)

17,servo ALM (0 open,1 close)

It is the type that system check ALM of servo drive(Pin12 in CN5),1 mean normal close,0 open.

18,SP ALM1 (0 open,1 close)

It is the type that system check ALM1 of Spindle(Pin5 in CN3),1 mean normal close,0 open.

19,Tool ALM2 (0 open,1 close)

It is the type that system check ALM2 of tool(Pin2 in CN10),1 mean normal close,0 open.

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

It is for set the chuck control signal is single or double,M10/M71 are the two signal.

When set as 0,only use M10 to control chuck.when M10 is valid,tighten tool,otherwise invalid.

When set as 1,use M10&M71 to control chuck.when M10 is valid and M71 is invalid,tighten tool,otherwise when M10 is invalid and M71 is valid,loose tool.(M10 output M10,M11 output M71)

Check whether it is in place,P40 in Axis parameter,1 means check,0 means no. When set as check,M10 is in place,and the system also check whether M40 is valid automatically,when M40 is valid,do next step.Loose tool M11,tool is in place,and the system also check whether M12 is valid automatically,when M12 is valid,do next step.

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

22,Outside chuck control(0 no,1 yes)

It is for whether use external switch(button) to control tighten/loose tool,it is reciprocating signal,one time is valid,tighten tool,next time is invalid,loose tool.when set 0 means without external switch,when with external switch,the signal

is M16.

23,Outside finial control(0 no,1 yes)

24,M10M11 short signal time(s)

It is for stay time that output M10,M71 is short signal.0 means them is long signal.unit:s.

25,M79M78 short signal time(s)

26,Emerge Stop(0 open,1 close)

It is the type of emerge stop1-switch in operation panel.for safe,advice set 1--Normal close.

27,Emerge Stop2(0 open,1 close)

It is the type of emerge stop2-switch in handwheel or external switch.(Pin5 in CN11).for safe,advice set 1--Normal close.

28,Run status output M69 STOP output M65(0 invalid,1 valid)

29,Alarm status output M67(0 invalid,1 valid)

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

31,Is enable I/O PLC program

32,Is enable High speed I/O PLC program

33,HY make run signal :[1 mean Yes,0 mean No]

34,HA make stop signal :[1 mean Yes,0 mean No]

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

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

37,Velocitv of RS232

[0=7200; 1=9600; 2=14400; 3=19200; 4=38400; 5=57600; 6=115200]

Note: the bigger the value is,the unstabler it is.

The bits of both ends must keep same.

38,Lock Manual rampit func key :[8 Yes]

39,Special parameter

40,Special parameter

41,Bake current parameter

It is defined to ex-factory value set.it is used for bake current parameter after test system well.

42,Resume original parameter

601,Make current to Step Motor Parameter

602,Make current to Servo Motor Parameter

3.6.6 Work coordinate parameter

This parameter coordinate multiple functions, namely six workpiece coordinate system and a machine coordinate system G53. A machining program can set a

workpiece coordinate system can also be set up multiple workpiece coordinate system, the workpiece coordinate system can be changed to move its origin. That is the value of the parameter in the coordinates of its own coordinate origin (zero) coordinate value in the machine coordinate system.

G54 to G59 can be set with six workpiece coordinate system, the coordinate system settings interface can be modified 6 origin of the workpiece coordinate system coordinate value in the machine coordinate system.

- 1,X of work coordinates G54(mm)
- 2,C(Y) of work coordinates G54(mm)
- 3,Z of work coordinates G54(mm)
- 4,A of work coordinates G54(mm)
- 6,X of work coordinates G55(mm)
- 7,C(Y) of work coordinates G55(mm)
- 8,Z of work coordinates G55(mm)
- 9,A of work coordinates G55(mm)
- 11,X of work coordinates G56(mm)
- 12,C(Y) of work coordinates G56(mm)
- 13,Z of work coordinates G56(mm)
- 14,A of work coordinates G56(mm)
- 16,X of work coordinates G57(mm)
- 17,C(Y) of work coordinates G57(mm)
- 18,Z of work coordinates G57(mm)
- 19,A of work coordinates G57(mm)
- 21,X of work coordinates G58(mm)
- 22,C(Y) of work coordinates G58(mm)
- 23,Z of work coordinates G58(mm)
- 24,A of work coordinates G58(mm)
- 26,X of work coordinates G59(mm)
- 27,C(Y) of work coordinates G59(mm)
- 28,Z of work coordinates G59(mm)
- 29-1,How to set up the workpiece coordinate system?

- a).press “F8-coordinate”key,select related workpiece coordinate system(54-59)
- b).move machine to suitable position that easy to measure in manual,measured the related coordinate value between this point (zero point in the workpiece)to Home of G53.
- c).press “F7-set coordinate”,press “X”key and enter,insert the value to dialog,and enter.
- d).press “F7-set coordinate”,press “Z”key and enter,insert the value to dialog,and enter.

It is finished now,enter coordinate screen in parameter,it could be seen the parameter values have been set well,it is offset value between workpiece coordinate and machine coordinate.

- 29-2,How to adjust the offset value?

If set up workpiece coordinate system well,when it needs to adjust the offset

value,it could be set by enter the coordinate parameter screen,steps is as follow:

In the coordinate parameter screen,selected the parameter,press “Enter”,and pop up dialog,input the offset value(also Increments,example:offset 10mm in negative direction,input -10,it is okay),press “Enter”.

Explanation:1.when the parameter is altered well,the coordinate main screen will refresh the corresponding coordinate value soon.

2.brackets in these parameters,it means the sum ,which is offset or adjust every time.It is suitable to look for the offset every time.

3.6.7 Password

The password is order to avoid modified accidentally and ensure the system work in normal condition.The system adopt three permissions, “CNC Factory”, “Machine Factory” and “User”.

The original condition is “CNC factory” is set, “Machine factory” and “User” isn’t set.

After set new password(set new password ,it need original password),please remember the new password ,and the original password wasn’t work.

Note:the password must be 6 bit data,the data could be number and letter.

password setting include:

1,Is enable CNC Co.’s password ?

It is for inner parameter,it couldn’t be set.

2,Is enable Machine Co.’s password ?

It is for whether display and set the parameter that is related to machine’s configuration.Its original password is “NEWNEW”.

3,Is enable User’s password ?

It is for whether display and set the parameter that is related to processing.Its original password is “KERKER”.

4,Modify CNC Co.’s password:

5,Modify Machine Co.’s password:

6,Modify User’s password:

7,curry word time: (days)

8,Version of Software.

3.7 Set parameter of tool redeem

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.

3.7.1 Radius compensation

Man Con		N00000		2013-08-23 11:45																																																																	
<table border="1"> <tr><td>01</td><td>X:</td><td>0.000</td><td>Z:</td><td>0.000</td><td>C:</td><td>0.000</td><td>AB</td></tr> <tr><td>02</td><td>X:</td><td>0.000</td><td>Z:</td><td>0.000</td><td>C:</td><td>0.000</td><td>AB</td></tr> <tr><td>03</td><td>X:</td><td>0.000</td><td>Z:</td><td>0.000</td><td>C:</td><td>0.000</td><td>AB</td></tr> <tr><td>04</td><td>X:</td><td>0.000</td><td>Z:</td><td>0.000</td><td>C:</td><td>0.000</td><td>AB</td></tr> <tr><td>05</td><td>X:</td><td>0.000</td><td>Z:</td><td>0.000</td><td>C:</td><td>0.000</td><td>AB</td></tr> <tr><td>06</td><td>X:</td><td>0.000</td><td>Z:</td><td>0.000</td><td>C:</td><td>0.000</td><td>AB</td></tr> <tr><td>07</td><td>X:</td><td>0.000</td><td>Z:</td><td>0.000</td><td>C:</td><td>0.000</td><td>AB</td></tr> <tr><td>08</td><td>X:</td><td>0.000</td><td>Z:</td><td>0.000</td><td>C:</td><td>0.000</td><td>AB</td></tr> </table>				01	X:	0.000	Z:	0.000	C:	0.000	AB	02	X:	0.000	Z:	0.000	C:	0.000	AB	03	X:	0.000	Z:	0.000	C:	0.000	AB	04	X:	0.000	Z:	0.000	C:	0.000	AB	05	X:	0.000	Z:	0.000	C:	0.000	AB	06	X:	0.000	Z:	0.000	C:	0.000	AB	07	X:	0.000	Z:	0.000	C:	0.000	AB	08	X:	0.000	Z:	0.000	C:	0.000	AB	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 SPpm 0	
01	X:	0.000	Z:	0.000	C:	0.000	AB																																																														
02	X:	0.000	Z:	0.000	C:	0.000	AB																																																														
03	X:	0.000	Z:	0.000	C:	0.000	AB																																																														
04	X:	0.000	Z:	0.000	C:	0.000	AB																																																														
05	X:	0.000	Z:	0.000	C:	0.000	AB																																																														
06	X:	0.000	Z:	0.000	C:	0.000	AB																																																														
07	X:	0.000	Z:	0.000	C:	0.000	AB																																																														
08	X:	0.000	Z:	0.000	C:	0.000	AB																																																														
No Alarm																																																																					
Radius		Length		ACLEA CLEAR SetTool ToolPait Set CANCEL																																																																	

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 popup 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.

3.7.2 Length of redeem

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 workpiece 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.

3.7.3 Posit tool

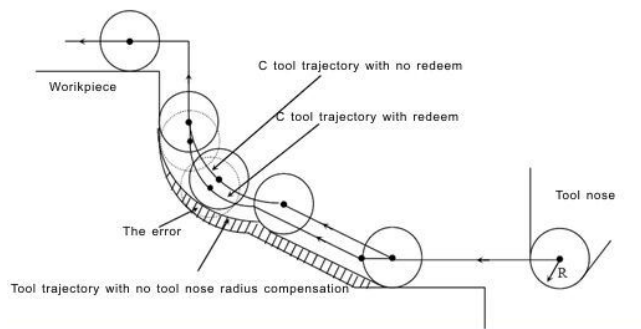
Press “F6” to enter posit tool interface in redeem interface. The parameter is used to set the kind of tool when adopting radius compensation of tool.

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.

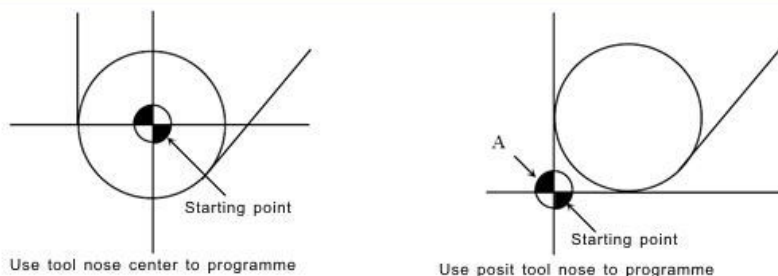
Introduction of posit tool

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.



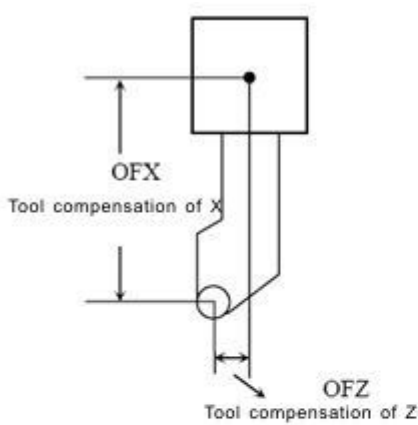
A. Posit tool

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 programmed do not need to consider the tool radius. The tool in the origin of the position relation as shown below.

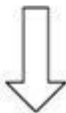


Pay attention: For some lathes with electrical datum point, a standard point(like center nose of tool) can be the starting point. The distance form the standard point to the radius center of tool rest or posit tool can be the offset of tool.

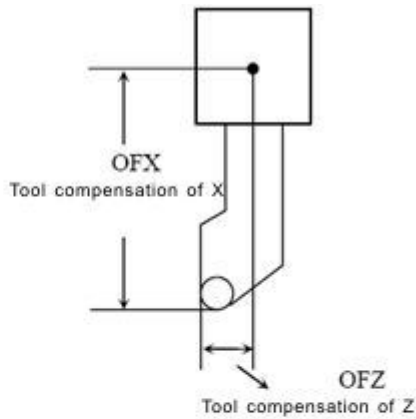
When tool rest is at the beginning point.



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



Put the starting point on tool nose center



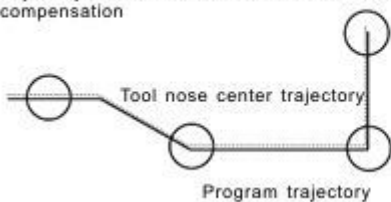
Set the distance from standard point to tool nose as the compensation



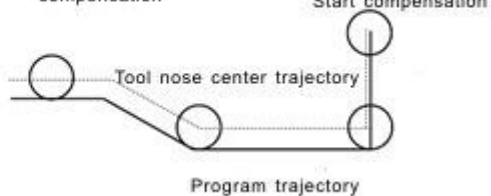
Put the starting point on posit tool nose

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

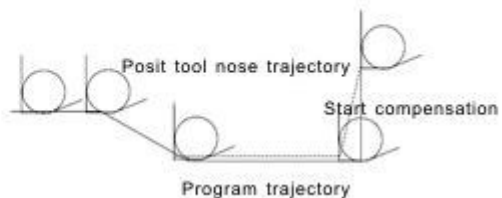


Using the posit tool to programme

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

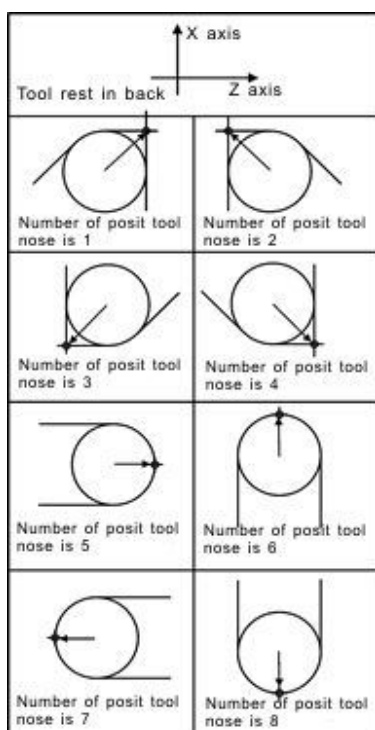


It will cut correctly if there is tool nose radius compensation

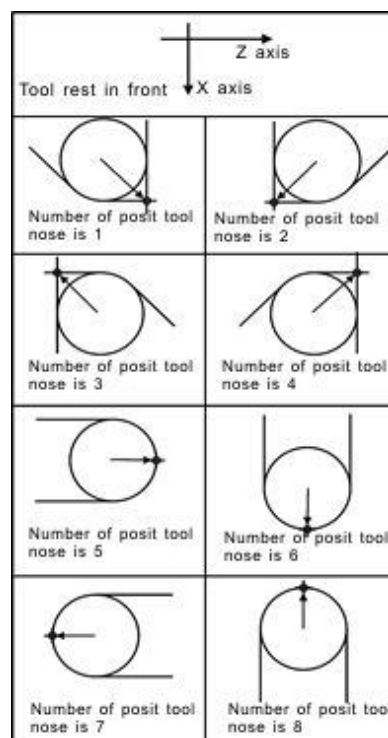


B. Direction of posit tool

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.

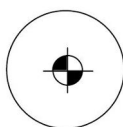


The direction of posit tool of backward tool rest



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.



3.7.4 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.

3.8 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 Milling. The system adopts storage mode of screw compensation: Making the Milling’s datum 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		2013-04-03 16:08	
Basic Parameter		No. Coord(mm) Value(um)		Program 02171.NC	
1, Standby: 0		1) <160.000> 0		Instruction code	
2, Neg-point: 3		2) <80.000> 0		G54	
3, Pos-point: 2		3) <0.000> 0		T01H0D0	
4, multiple: 1.000		4) <-80.000> 0		Machine Status	
5, distance(um): 80000		5) <-160.000> 0		M05 M09 M10	
		6) <-240.000> 0		M78 M33 M70	
				G00 X100%	
				F500 X150%	
				S5000 X100%	
				Machine Coord	
				X 0.000	
				Y 0.000	
				Z 0.000	
N-X-axis	T-Y-axis	R-Z-axis	Q-A-axis	PartTime 0: 0	
A-	B-ChePro	C-CLEAR	D-CANCEL	PartNo 1	
No Alarm				SPrpm 0	

Screw compensation interface

Screw compensation by the axis as the unit to set storage, set X Y Z axis separately, by pressing “N” “T” “R” 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 value:

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.

Man Con		N00000		2014-07-19 10:30	
Basic Parameter 1, Standby: 0 2, Neg-point: 3 3, Pos-point: 2 4, multiple: 1.000 5, distance(um): 80000		No. Coord(mm) Value(um) 1)<160.000> 0 2)<80.000> 0 3)<0.000> 0 4)<-80.000> 0 5)<-160.000> 0 6)<-240.000> 0		Program G110 Instruction code G53 T02H0D0 Machine Status M05 M09 M10 M78 M33 M70 G00 X100% F250 X150% S0 X100% Machine Coord X 2.921 Y 0.000 Z -14.930 PartTime 0:3 PartNo 596 SPrpm 0	
5, distance(um): 80000					
N-X-axis	T-Y-axis	R-Z-axis	Q-		
A-	B-ChePro	C-CLEAR	D-CANCEL		
No Alarm					

Man Con		N00000		2014-07-19 10:30	
Basic Parameter 1, Standby: 0 2, Neg-point: 3 3, Pos-point: 2 4, multiple: 1.000 5, distance(um): 80000		No. Coord(mm) Value(um) 1)<160.000> 0 2)<80.000> 0 3)<0.000> 0 4)<-80.000> 0 5)<-160.000> 0 6)<-240.000> 0		Program G110 Instruction code G53 T02H0D0 Machine Status M05 M09 M10 M78 M33 M70 G00 X100% F250 X150% S0 X100% Machine Coord X 2.921 Y 0.000 Z -14.930 PartTime 0:3 PartNo 596 SPrpm 0	
5, distance(um): p					
N-X-axis	T-Y-axis	R-Z-axis	Q-		
A-	B-ChePro	C-CLEAR	D-CANCEL		
No Alarm					

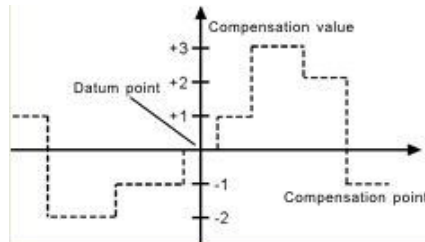
Test program generation automatically

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 datum point).

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

For example:

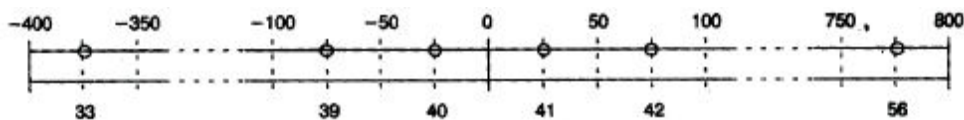
Example 1: Linear axis: when length of travel is -400mm~+800mm, spacing of points is 50mm, reference point compensation is No.40, it can figure out that Compensation point of farthest end in negative direction is:

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

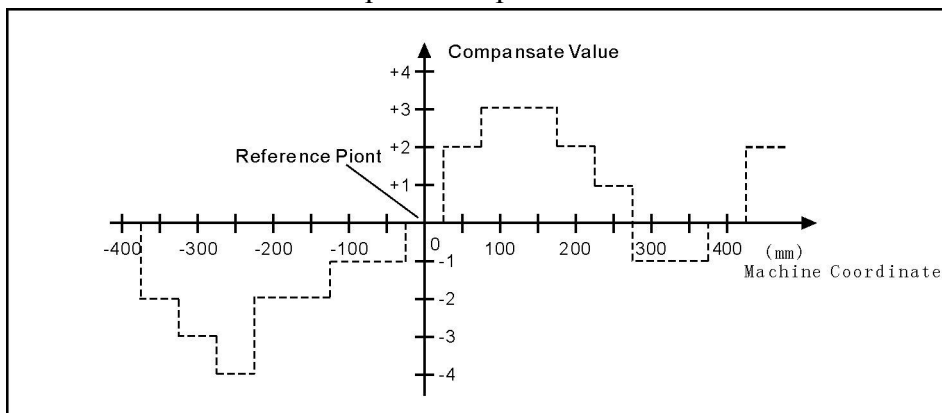
Compensation point interval: 50000

Corresponding compensation point and value:

The compensation value in corresponding compensation point:

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

The contrasted chart of compensation points and value as follows:

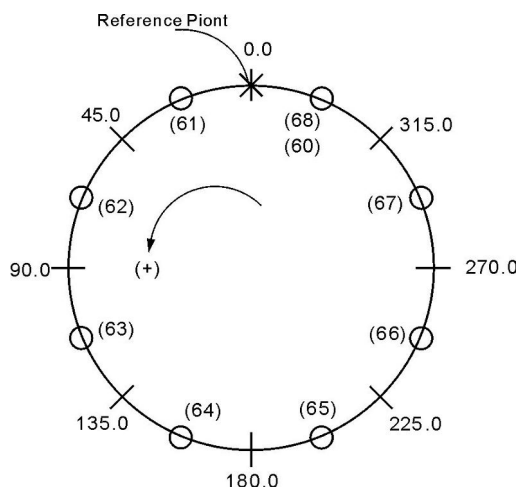


Example 2: rotor axis: when movement per revolution is 360° , interval of points 45° , reference point compensation NO. 60, Compensation point NO. of farthest end in negative direction is usually same as reference point compensation point NO.

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:

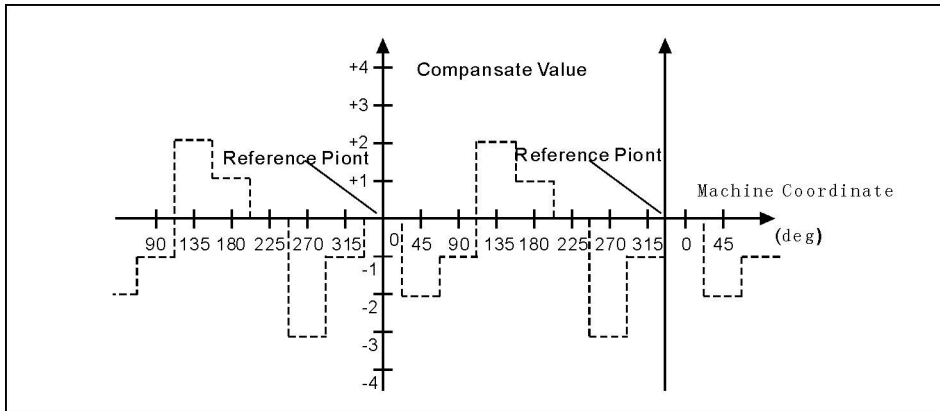


Note: input value in small circle. If the total amount from 61 to 68 doesn't equal 0, accumulated pitch error per revolution will deviate, so same value shall be put in 60 and 68.

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:



3.9 Input/output diagnosis

Press “Diagnosis” to enter the diagnosis interface in parameter interface.

Press “T” and “Pgup Pgdn” to check the status of input and output, press “Q” to check alarm information.

System diagnosis interface(Input signal)

Man Con		N00000		2012-12-26 19:28	
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 X20	X17 Z20
X18 KLEFT	X19 KRIGHT	X20 STOP	X21 TOK	X22 ALM	X23 ALM1
X24 ALM2	X25 M28	X26 M24	X27 M22	X28 M18	X29 M12
X30 M14	X31 M16	X32 HX/DS8	X33 HY/DS4	X34 HZ/DS2	X35 HA/DS0
X36 HX1/K8	X37 HX10/K	X38 HX100/	X39 HOFF/K		
Program 99.TXT Instruction code G53 T0302 Machine Status M05 M09 M10 M78 M33 M70 G00 X100% F6000 X100% S600 X100% Machine Coord X 0.000 Z 0.000 C 0.000					
N-	J - I/O	V-	Q - ALARM	PartTime	0: 37
A-	B - Reset	C-	D - CANCEL	PartNo	231
No Alarm				SPrpm	0

System diagnosis interface(output signal)

Man Con		N00000		2012-12-26 19:29	
Output Point					
Y00 M61	Y01 M63	Y02 M65	Y03 M67	Y04 M69	Y05 M71
Y06 M73	Y07 M75	Y08 M32	Y09 M79	Y10 M10	Y11 M08
Y12 M05	Y13 M04	Y14 M03	Y15 M75	Y16 LRUN	Y17 INTH
Y18 +T	Y19 -T	Y20 S04	Y21 S03	Y22 S02	Y23 S01
N- J - I/O V- Q - ALARM A- B - Reset C- D - CANCEL No Alarm					
Program 99.TXT Instruction code G53 T0302 Machine Status M05 M09 M10 M78 M33 M70 G00 X100% F6000 X100% S600 X100% Machine Coord X 0.000 Z 0.000 C 0.000 PartTime 0: 37 PartNo 231 SPrpm 0					

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		2014-07-19 10:33	
Clear history press DEL key, Reset Alarm pre					
Spindle encode check: 0 (No 1 encode)					
1,[Curent Alarm]: NO					
2,[History Alarm](13h46m): Emergent Stop					
N- T - I/O R- Q - ALARM A- B - Reset C- D - CANCEL No Alarm					
Program G110 Instruction code G53 T02H0D0 Machine Status M05 M09 M10 M78 M33 M70 G00 X100% F250 X150% S0 X100% Machine Coord X 2.921 Y 0.000 Z -14.930 PartTime 0: 3 PartNo 596 SPrpm 0					

Alarm information interface

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

3.10 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		2013-03-28 08:56	
File/File folde in system			Program 02171.NC		
CurDir: /NC			Instruction code		
02171.NC			G54		
1000.TXT			T01H0D0		
1000X			Machine Status		
1001			M05 M09 M10		
1002			M78 M33 M70		
120			G00 X100%		
1201			F500 X150%		
121			S5000 X100%		
1234			Machine Coord		
147			X 0.000		
22			Y 0.000		
84			Z 0.000		
841			PartTime 0: 0		
98			PartNo 1		
Compile-P: Receive-J, Tansmit-K, DNC-L			SPrpm 0		
N-New/Sek	T-COPY	R-RENAH	Q-INFOR		
A-LAST	B-USBdisk	C-EXEC	D-CANCEL		
No Alarm					

Center part of screen for program display, current program is showed by reverse display, move PgUp、PgDn to choose program, and then press“Enter”to edit current program. Functional keys“N、T、R、Q、A、B、C、D” include: “new file/search”、 “copy”、 “rename”、 “information”、 “last grade”“USB disc”、 “execute program”、 “cancel”.

3.10.1 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.

The file name and the Chinese input of program content:

The directory interface of the "new / search", "copy", "rename", "to copy into the system", "to copy into the USB" can enter the Chinese characters; change the input method according to the prompt of system. Also can input Chinese characters in

program editing interface according to the prompt of system.

Man Con		N00000		2013-04-25 14:05	
/NC/%99.TXT			Program %99.TXT		
N1 M03 S500			Instruction code		
N2 G00 X200 Z200			G54		
N3 G00 U-44			T01H0D0		
N4 W-5			Machine Status		
N5 G01 U4 W-2 F300			M05 M09 M10		
N6 W-11			M78 M33 M70		
N7 U-4 W-2			G00 X100%		
N8 W-5			F500 X150%		
N9 U10 W-10			S2700 X100%		
N10 W-6			Machine Coord		
N11 G02 U-6 W-9 I12 K-9			X 0.000		
N12 U10 W-15 I25 K0			Y 0.000		
			Z 0.000		
Press Rapit Key			No.1 /Total27		
N-COMPIL		T-FrELine		R-TeachIn	
A-		B-DelLine		Q-POS	
		C->>		D-CANCEL	
No Alarm					
PartTime			0:0		
PartNo			2		
SPrpm			0		

The screen prompt the editing program name at the top left corner in the editing status; The left is the content, the right is the information for milling status, the operation in the editing status as follows:

(1) The current cursor locate:

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

Press "Pgup" to the last page.

Press "Pgdn" to the next page.

(2) Character modification: Delete the character at the position of the cursor, then enter the new character.

(3) The character insertion: Enter a new direct character at the cursor position. When the input is the letter,the letter in front of automatically generating space. If you want to enter a space, first enter a letter, and then delete this letter.

(4) The character deletion: Press "Del" directly at the cursor position

(5) Inset the line: Press "Enter" directly, inset a line in front of the current line if the cursor is at the first line, otherwise insert a line after the current line.

(6) "Fast" superposition key operation:

The first function:

A, “”+“N”: compile the program.

B, “”+“J”: to the fist line or last line of the program.

C, “”+“V”: teaching function, enter the handwheel status; Press “Q” to read the tool coordinate in the current workpiece coordinate system according to the corresponding axis X/Y/Z/A (all the axis).

D, “”+“Q”: located to the specified line.

E, “”+“A”: no operation.

F, “”+“B”: delete the current line.

G, “”+“C”: the first or second page selection.

H, “”+“D”: Chinese characters and character input conversion.

The second function:

A, “”+“N”: delete the program block.

B, “”+“J”: copy the specified program block.

C, “”+“V”: sort the program.

D, “”+“Q”: to find the specified character.

E, “”+“A”: replacing the specified character.

F, “”+“B”: all the content to be replaced by the specified character.

G, “”+“C”: the first or second page selection.

H, “”+“D”: Chinese characters and character input conversion.

3.10.2 Copy

Press “↑ ↓” in program main interface to select program which need to copy and press “T” to pop up 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.

3.10.3 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.

3.10.4 Rename

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

3.10.5 Information

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

3.10.6 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.

3.10.7 Folder management

You can build a file in this system, Press “N” 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 “A” go to the last folder.

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

3.10.8 Select automatical program to run

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

3.10.9 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 “J” 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 “K” 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.

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

A. The processing program management

Copy the files or folder of U-disk into system

After connecting the U-disk, press “B” to enter the U-disk directory in program main interface. Press “↑ ↓” to move cursor to select file or folder to copy and press “Q” 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.

Press “J” to copy all the program in USB into system.

Copy the files or folder of system into U-disk

Press “↑ ↓” to move cursor to select file or folder and press “B”, press “R” 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.

Press “K” to copy all the program in system into USB.

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 “B” 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 “T” to import code when appearing the files and press “Enter” to confirm, wait for seconds to copied the parameter successfully. Press “B” 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 “B” to show the files in U-disk. Press “N” 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 “B” 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.

Chapter4 Programming

Programming refers to process of using cnc language to describe machining track and actions based on the machining blueprint and technique requirement.

4.1 basic concepts

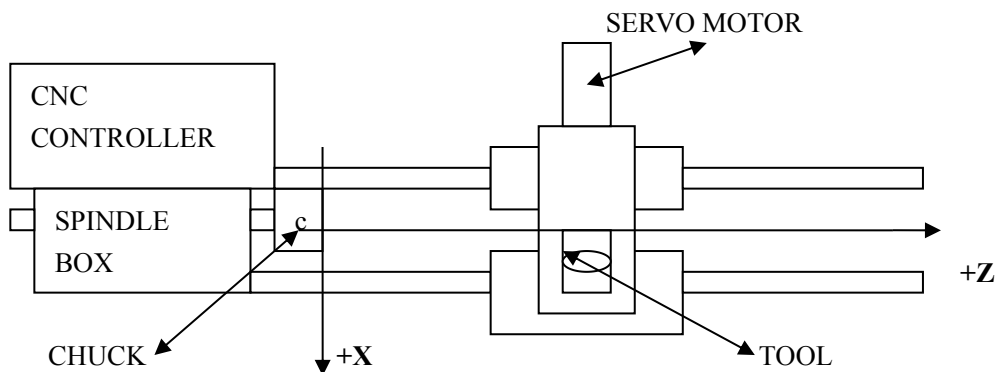
program segment: is a complete command line consisted of instruction segment and data segment.

Program: is a congregation of program segment by machining logic structure in order to complete the machining of workpiece.

Machine coordinate system: see fig Lathe top view.

Machine coordinate stipulates machine coordinate and direction, z axis parallels spindle, direction away from spindle is positive. X axis is vertical to spindle, the opposite of knifepoint is positive.

Lathe top view



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

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

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

Diameter programming: all X axis coordinate are presented by diameter.

Starting point: the place where the tool begins to move when program starts.

Name of program: the name of machining program.

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.

Suspending mode (destroying mode): is the instruction which can turn mode instruction into mode origin or destroy the mode. Such as M20 (program ending instruction), meaning the end of operation and returning to original status.

Non mode instruction: is the instruction which has no function to store, and only works in the segment of program.

4.2 Program instruction

4.2.1 Functional meaning of address symbol, data list.

Functions	Address symbol	Meaning	Data range
Program segment No	N	No of program segment	
Preparation function	G	Content and mode of designalated instruction operation	00-99
Auxiliary function	M	Auxiliary operation instruction	00-99
Tool function	T	Tool changing instruction	Tool T0101-9999
Spindle function	S	The first spindle speed	0-65000
Spindle function	SS	The second spindle speed	0-65000
Cutting speed	F	Speed per minute, per rotation	1-30000mm/min, 0.001-9.999mm/n
Coordinates character	XU ZW	The coordinates value of X Z axes	±99999.9999mm
Lead of screw thread	F(I)	F for metric, I for imperial	0.1-1000mm, 1-99teeth/inch
Core coordinates	IK	X Z axes coordinate increment value	±99999.9999mm
Arc radius	R	Arc radius value, tool radius value	0.001-99999.999mm
Delay time	XUP	Delay time of designalated delay	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	1-65000 1-99
Program skip	/	There is “/”befor N ,this line does not run.	

4.2.2 Program instruction table

Group	Nature	Code	Functions	Origin	Mode	remarks
1	Main control functions	G00		√	√	
		G01	Rapid point positioning		√	
		G02	Linear interpolation		√	
		G03	Circular—clockwise		√	
			Circular—counterclockwise		√	
		G32	spiral interpolation			
		G2	G02 spiral			
		G33	G03 spiral			
		G3	Check jump No alm			
		G31	Check jump alm			
	Loop comand	G70			√	
		G71	Finish manching loop		√	
		G72	Cylindrical face thick loop		√	
		G73	end face thick loop		√	
		G74	close loop		√	
			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	loop end		√	
		G80	cancel loop		√	
		0				

	Go start	G26 G26 1 G26 2 G26 3 G26 4	Z、X go start X go start Y go start Z go start A go start			
		G61 G61 1 G61 2 G61 3 G61 4	XZ Go G25 X Go G25 Y Go G25 Z Go G25 A Go G25			
	Save	G25	Save current coor		√	
	Go home	G28 G28 1 G28 2 G28 3 G28 4 M80 0	XZ go home X go home Y/C go home Z go home A go home C go encode Zero		√	
		G50	Setup coor system		√	
		G52	Setup part coor ststem		√	
		G18 4	Setup current Tool coor			
	set	G18 5	Setup all Tool coor			
2	lin e	G96	Constant line speed cutting		√	

	feed	G97	cancel	√	√	
		G98	Set feed per minute	√	√	
		G99	Set feed per revolution		√	
	mode	G15	cancel	√	√	
		G16	bally coor program		√	
		G21	metric program	√	√	
		G20	imperial		√	
3	delay	G04	Programmed dwell			
4	cut	G60	Nicety stop	√	√	
		G64	Continuum part		√	
5	compensation	G40	Cancel cutter compensation	√	√	
		G41	Cutter in the left of workpiece		√	
		G42	Cutter in the right of workpiece		√	
6	Work coor	G53	Machine coor	√	√	
		G54	Work coor1		√	
		G55	Work coor 2		√	
		G56	Work coor 3		√	
		G57	Work coor 4		√	
		G58	Work coor 5		√	
		G59	Work coor 6		√	
7	SP	M0 3	Spindle on clockwise		√	
		M0 4	Spindle on counterclockwise		√	
		M0 5	Spindle off	√	√	
8	Cooling	M0 8	Coolant on		√	
		M0 9	Coolant off	√	√	

9	Chuck	M10 M11	Chuck tightens Chuck looses	√	√ √	
10	Tailstock	M79 M78	Tighten Loose	√	√ √	M79
11	Lubrication	M32 M33	Lubrication on Lubrication off	√	√ √	M32
12	Huff	M59 M58	Huff on Huff off	√	√ √	M59 output
12	user-define output	M61 M60	user-define1		√ √	M61 output
		M63 M62	user-define2		√ √	M63 output
		M65 M64	user-define3		√ √	M65 output
		M67 M66	user-define4		√ √	M67 output
		M69 M68	user-define5		√ √	M69 output
		M71 M70	user-define6		√ √	M71 output

		M7 3	user-definde7		√	M73 output
		M7 2			√	
13	user-definde input	M7 5	user-definde8		√	M75 output
		M7 4			√	
		M1 2	Check M12 valid		√	M12 INPUT
		M1 3	Check M12 invalidate		√	
		M1 4	Check M14 valid		√	
		M1 5	Check M14 invalidate		√	
		M1 6	Check M16 valid		√	
		M1 7	Check M16 invalidate		√	
		M1 8	Check M18 valid		√	
		M1 9	Check M18 invalidate		√	
		M2 8	Check M28 valid		√	
		M2 9	Check M28 invalidate		√	
		M2 2	Check M22 valid		√	
		M2 3	Check M22 invalidate		√	
		M2 4	Check M24 valid		√	
		M2 5	Check M24 invalidate		√	

14	SP Gear	M4 1 M4 2 M4 3 M4 4	first gear second gear third gear fourth gear			
15	jump	M9 7 M9 8 M9 9	jump call return			
16	Program	M0 0 M0 1 M0 2 M3 0 M2 0	pause M22 avail pause Program end M05、 M09 end Loop go start			
17	SP speed	S SS	First SP Second SP		√ √	S=0-65000 SS=0-65000
18	Tool	Tab	A:xx B:xx		√	A、 b=00-12

4.3 Preparation functions

4.3.1 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, no irrelative parameters and operation in a segment.
- 5, “0” 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

4.3.2 Function of G Instructions

(1) Rapid motion(G00)

Tool move to instructive position according to G00 speed in parameter.

As absolute method, use section end point coordinate to program;

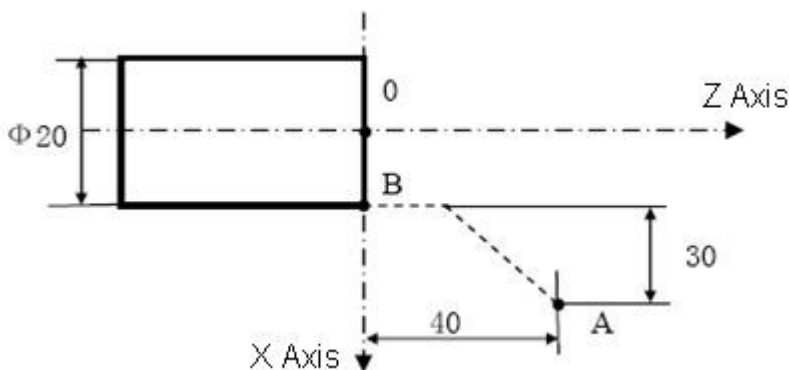
As increase method, use motion distance to program.

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

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 programming, calculation by programming when using relative coordinate;

G00 of every axis is set by parameter, the specified feeding speed with F is invalid. The speed rate 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.

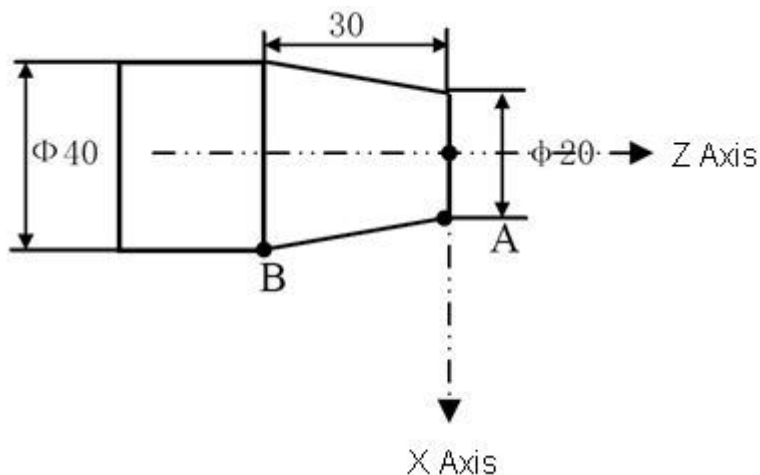
Format: G01 X/U- Y/V- Z/W- A- B- F- (Mode)

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.

Example: from A to B.

Absolute program: G01 X40 Z-30 F100

Relative program: G01 U20 W-30 F100



G01 instruction can also specify a mobile X-axis or Z-axis separately.

G01 is F feed rate can be motivated by the panel to override adjusted up or down to adjust the range (0% -150%).

G01 instruction can also be directly written G1.

(3) Arc interpolation(G02/G03)

Format: G02 X (U) __ Z (W) __ I__ K__ F__;

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

G02 X (U) __ Z (W) __ R__ F__;

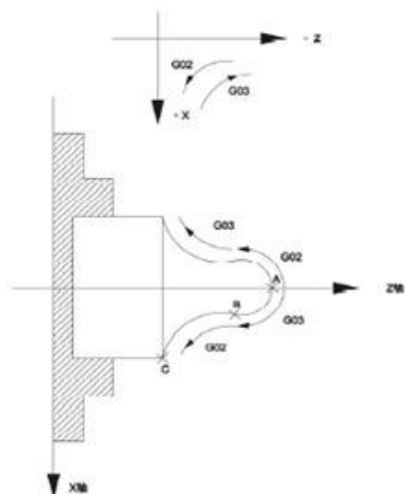
G03 X (U) __ Z (W) __ R__ F__;

Ranges I, K, R's: -99999.999 ~ + 99999.999.

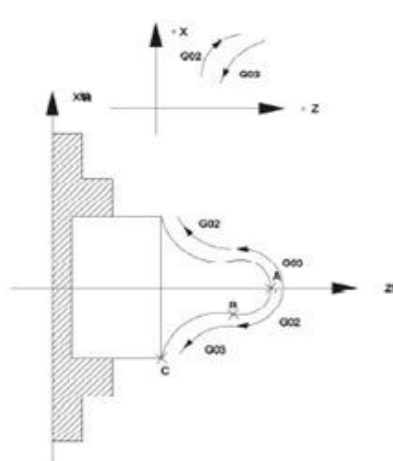
No.	Explanation		Code	
1	Direction of Circle		G02	Clockwise-CW
			G03	Counterclockwise-CCW
2	End position	Absolute value	X/Z	End position in coordinate
		Correspond value	U/W	Distance from start to end
3	Distance From start to center		I/K	I(radius) means X-axis,K means direction of Z-axis
	Radius		R	Radius(in radius)
4	Feeding speed		F	Speed along the arc tangent

G02 stands for Circular clockwise interpolation, and G03 for Circular counterclockwise interpolation. When tool system in different position,the direction is different.

Tool System in Front

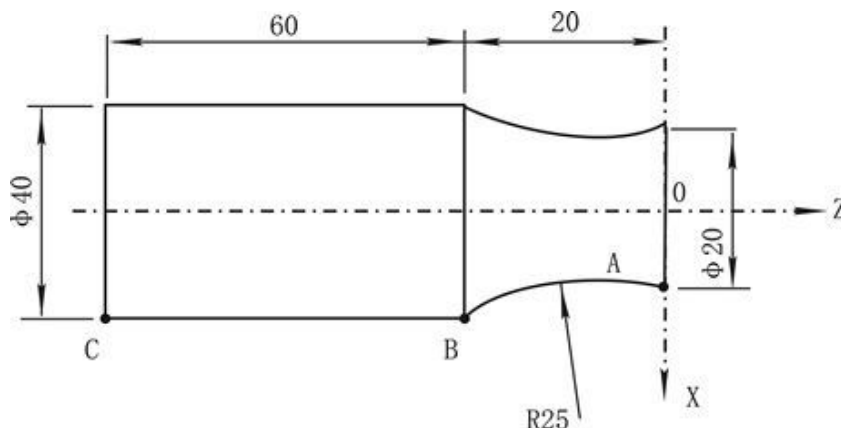


Tool System in Back



I is the 2 times of increment of X axis which is from 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 IK.



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 when; when I, K and R at the same time instruction, R effective, I, K 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 speed of the tool along the arc cutting direction.

(4) Screw thread (G32)

The system could be processed straight thread,tapper thread,straight thread continuous thread, end thread, metric thread, internal and external thread, trapezoidal thread, etc.

Straight thread: only input the direction and length of Z-axis;

Tapper thread: must input the direction and length of X-axis and Z-axis;

End thread(head face thread): only input the direction and length of X-axis;

Continuous thread: Using multi-stage instruction of G32;

Trapezoidal thread: offset the angle of SP in the second loop.

Variable lead thread : continuous input G32 command, enter a thread length of each program, lead of thread(F) is different , the second cycle will not begin to detect the encoder synchronization signal.

Format: G32 Z(W)_X(U)_ F(I)_ SP_

G32 is the spiral interpolation machining instruction. it is modal. X(U)_,Z(W)_ is the value of end point of thread in absolutely/correspond coordinate system, Z/W is the thread length of Z axis, it cuts strait thread; X/U is the diameter thread of X axis, it cuts head face thread;

F is metric lead(pitch), the amount of displacement of the long axis of the spindle revolution. Range is 0.1-1000 ;Unit:mm;

I is imperial lead(pitch), The number of turns of the spindle when long axis move one inch(25.4mm). Range is 0.1-99 ;Unit:teeth/inch;

Max Pitch=Lines of Spindle's encoder/50mm.

SP(Q) is angle of tool's cutting, Unit:degree(°), range:0-360°; Q ,Unit:0.001degree(°). Not specified the value of SP, its value is 0.

{L is multiple thread head numbers, range is 1-999, default value is 1.}

The use of thread machining instruction must be under the condition that machine has quipped with photoelectric encoder; otherwise the system is in the status of standby. When spindle rotates clockwise, it machining plus thread in the Z axis negative direction, and left-hand thread in positive direction. The cutting feed speed $F=K \times N$ is appropriate to the machining thread, over high speed will destroy the teeth, this system requires $N \leq 2000 \text{ n/min}$, $F \leq 3000 \text{ mm/min}$.

For example:

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

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

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

If rough or fine machining is needed, it can add fine machining rotating speed into thread fine machining instruction.

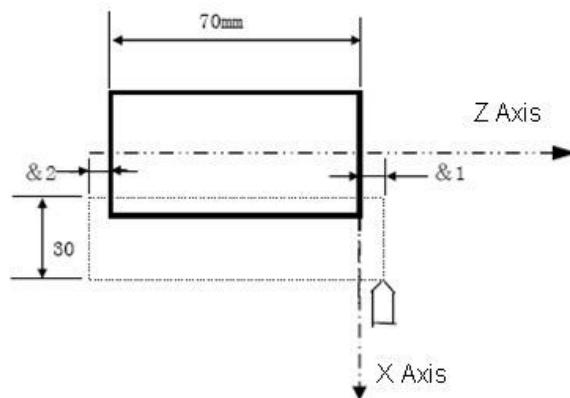
For example1:

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

N0010 G00 U-1 ; advance of tool

N0020 G32 W-30 K1.5 ; rough machining

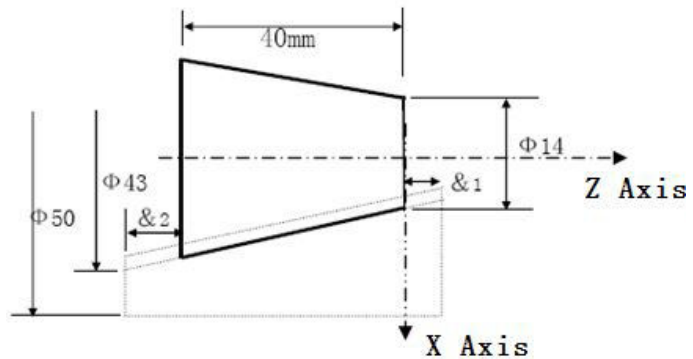
For example2: G32 cut threaded barrel: lead(pitch)=4mm, &1=3mm (Speed up the thread segments, &1 $\geq$ 3mm); &2=1.5mm (Thread deceleration segment, &2 $\geq$ 1.5mm). Upon request, cut in twice.



```
G00 U-62.0;
G32 W-74.5 F4.0;
G00 U62;
W74.5;
U-64;
G32 W-74.5;
G00 U64.0;
W74.5;
```

.....

For example3:



```
G00 X12 Z3.0;
G32 X41.0 Z-41.5 F3.5;
G00 X50;
Z3;
X10;
G32 X39 Z-41.5;
G00 X50;
Z3;
```

.....

Pay attention:

1. When the transmission of spindle and encoder not as 1:1, please modify the axis parameter No.412 No.413;
412 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);
413, 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.

(5) Circularity screw thread(G332、 G333)

Format: G332/G333 Z (W) - X (U) - R - F (I) - SP-
Use method refer G02、 G03、 G32 instruction.

(6) delay Instruction(G04)

Require of work process, delays some time before execute other motion.

Format:	G04	P_;
or	G04	X_;
Or	G04	U_;

Function: Every axis is stop and mode instruction is still working when carry out this instruction, after delaying the specified time to carry out the next program segment.

Instruction introduction:

a. The unit of P delay time is ms(Millisecond)/

b. The unit of X and U delay time are S.

c. Example:

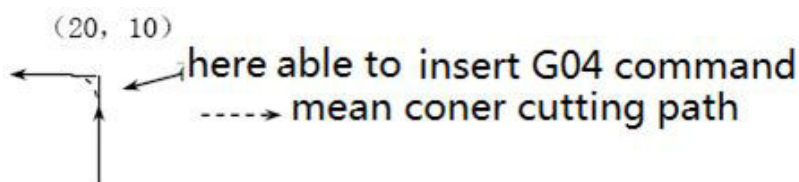
G04 X1; delay 1s.

G04 P1000; delay 1s.

G04 U1; delay 1s.

d. 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;

.....

Pay attention: Set No.21 parameter in “N”(processing) parameter to clear the over cutting.

(7) Return Reference (G28/G281/G282/G283/G284/G285 /M800 / M881)

Reference point is a particular position point in machine.If there is machine zero point,it is reference point, otherwise,set float zero point and take it as reference point.it could return reference point in Manual or in Auto.

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

format:	G28 X/U_ Y(C)/V_ Z/W_ A_ B_ ;ZXY return to reference
	G281 ;only X return to reference
	G282 ;only Y return to reference,
	G283 ;only Z return to reference
	G284 ;only A return to reference
	G285 ;only B return to reference

Note:when C axis return reference point,the system only detect the Y0 signal,not detect Z0 signal.

M800 C axis return zero point of encoder(Z0),when C is as rotation axis,output M75 signal, the spindle servo select position control mode,when system run M03/M04,turn off M75 signal.

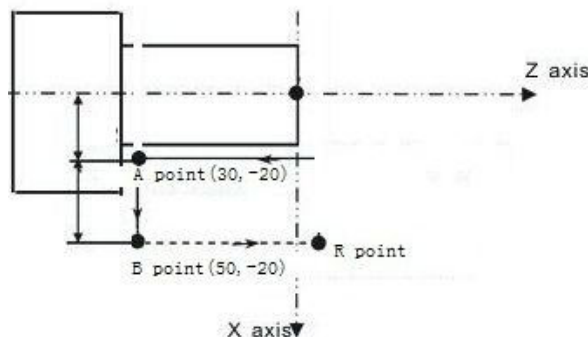
M881 C axis exact stop,send out Y31 signal,wait for answer signal X40,when X40 is in position,and then cancel Y31 signal.

G28 X/U_ Y(C)/V_ Z/W_ A_ B_ ;the coordinate value is middle point when return.

The steps is as follow picture:B pint is middle point,R point is reference point.

(1)quickly move from current position(A point) to middle point (B point)

(2) quickly move from middle point(B point) to reference point(R point)



Program: G28 X50 Z-20 ;

Or G28 U20 W0;

Pay attention: 1. After turning on the power, if not process back to datum point manually at a time. When the instruction is G28, the motion from middle point to datum point is the same as backing to datum point manually.

2. Backing to datum point when the instruction is G28, if just specify middle point of an axis, that's the axis is backing to the datum point, the other axis is not back.

(8) Setup workpiece coordinate system (G50)

Format: G50 X (x) Z (z) ;

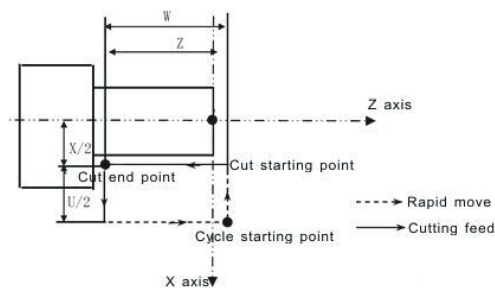
Pay attention: In the status of compensation, if use G50 to set the coordinate, then use G50 to set the position in process coordinate system. Usually cancel the tool compensation first before starting program. Automatically cancel the tool compensation after the system backing to the datum point.

G50 S; Set the maximum speed of spindle when cutting constant linear velocity.

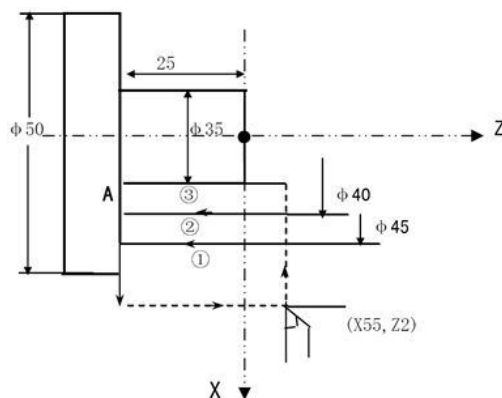
According to this instruction, set the maximum spindle speed of instruction G96 constant line speed in cutting.

(9) Column or taper loop(G90)

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



Column loop cutting



Usage of G90

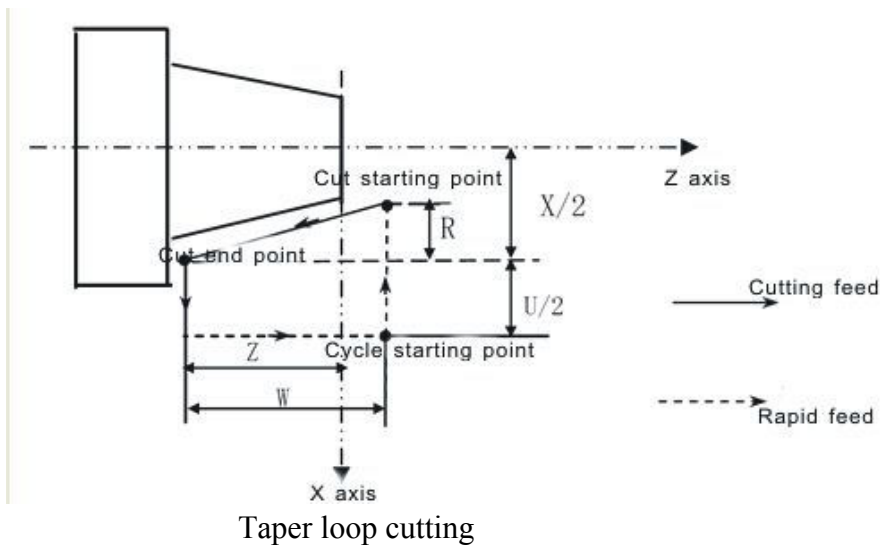
For example:

```
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 ;
```

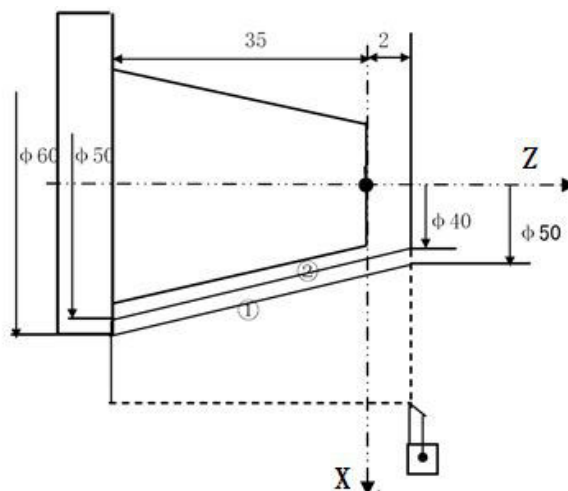
Taper loop format: G90 X(U)___ Z (W) ___ R___ F___;



For example:

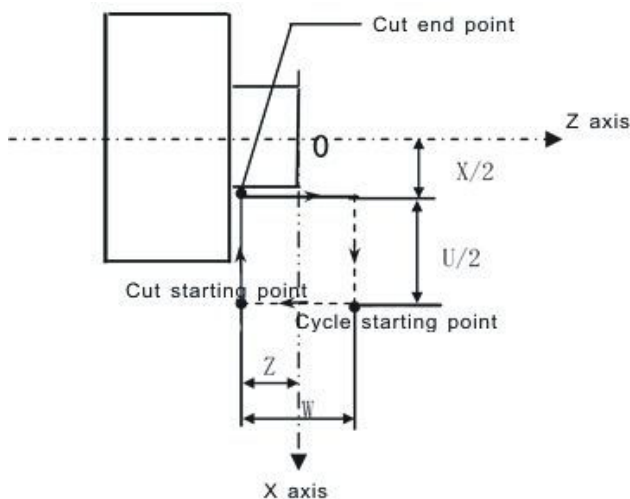
```

N10 M03 S1000;
N20 T0101;
N30 G00 X65 Z5;
N50 G96 S120;
N60 G99 G01 Z2 F1 M08;
N70 G90 X60 Z-35 R-5 F0.2;
N80 X50;
N90 G00 G98 X100 Z100 M09;
N100 G97 S1000 T0100;
N110 M05;
N120 M30;
    
```

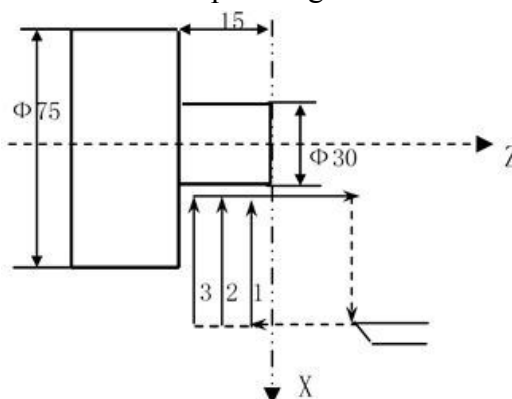


(10) End face loop(G94)

Format: G94 X(U)___ Z (W) ___ F___;



End face loop cutting



Usage of G94

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;

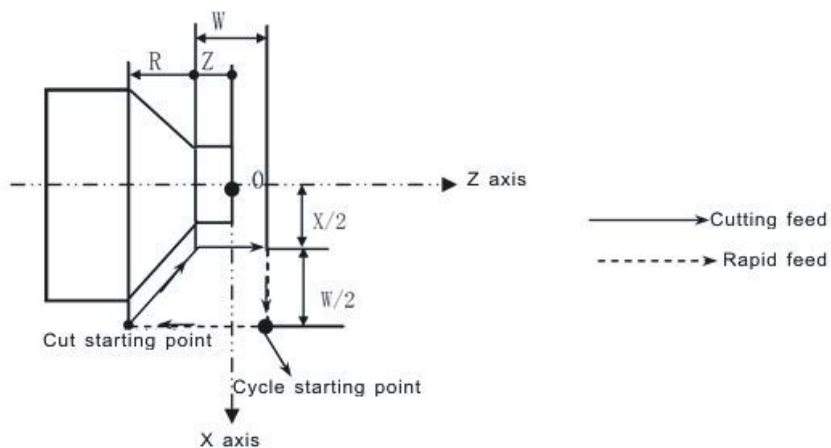
N70 G94 X30 Z-10;

N80 G00 Z-8;

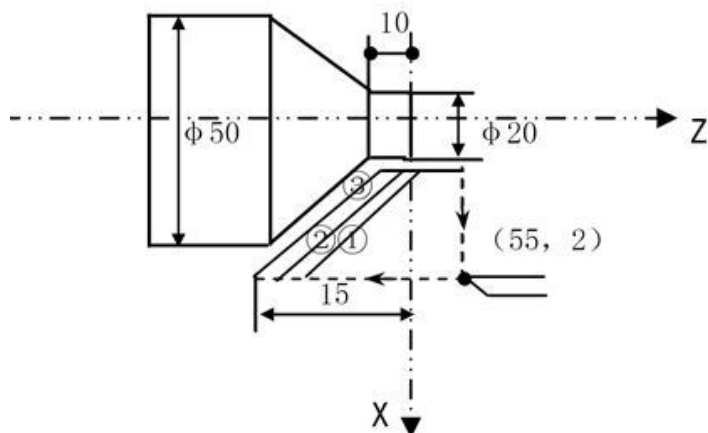
N90 G94 X30 Z-15;

N100 G00 X100 Z60;

Taper end face loop format: G94 X(U)___ Z (W) ___ R___ F___;



Taper end face loop cutting



For example:

.....

N40 G01 X55 Z2 F200;

N50 G94 X20 Z0 R-5 F100;

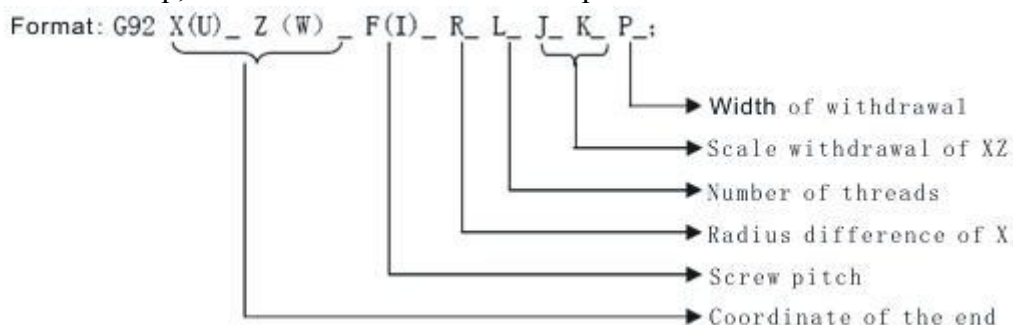
N60 Z-5;
N70 Z-10;
N80 G00 X Z;

.....

R-5 in N50 program: $R-5 = -15 - (-10) = -5\text{mm}$

(11) Screw thread loop (G92)

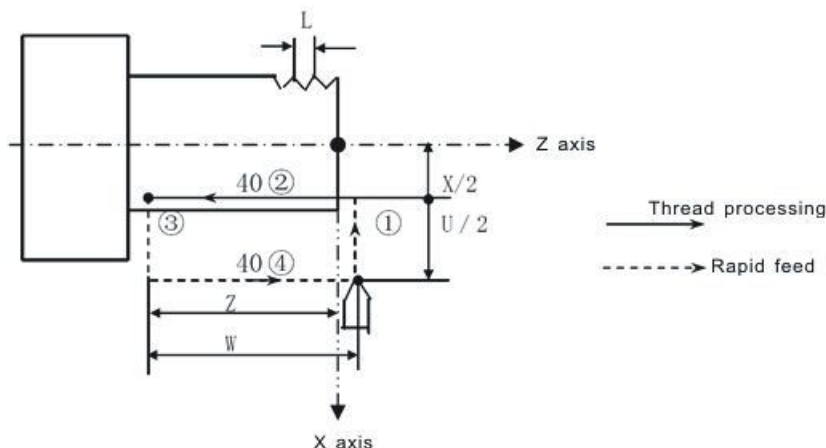
Use G92 instruction, you can thread cutting process, from initial point of departure "cut - cut thread - let the knife - return thread machining starting point," the four movements as a loop, with a block command to complete.



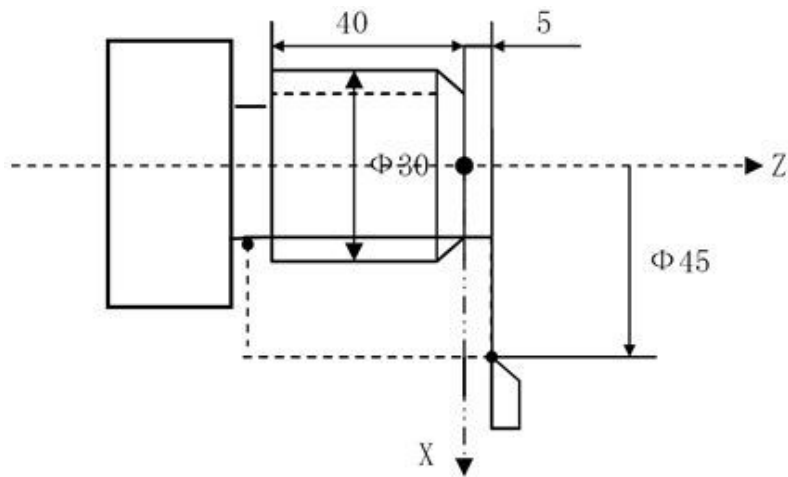
1) Straight screw thread loop format:

G92 X(U) _ Z (W) _ F/I _ ;

Cycle as shown in Fig, X, Z coordinate values of the thread cutting end point, U, W for the thread cutting end point relative to the coordinates of the cycle starting point of the component, there are plus and minus signs. F / I specify the thread lead L, designated with the G32.



Straight screw thread loop



Program:

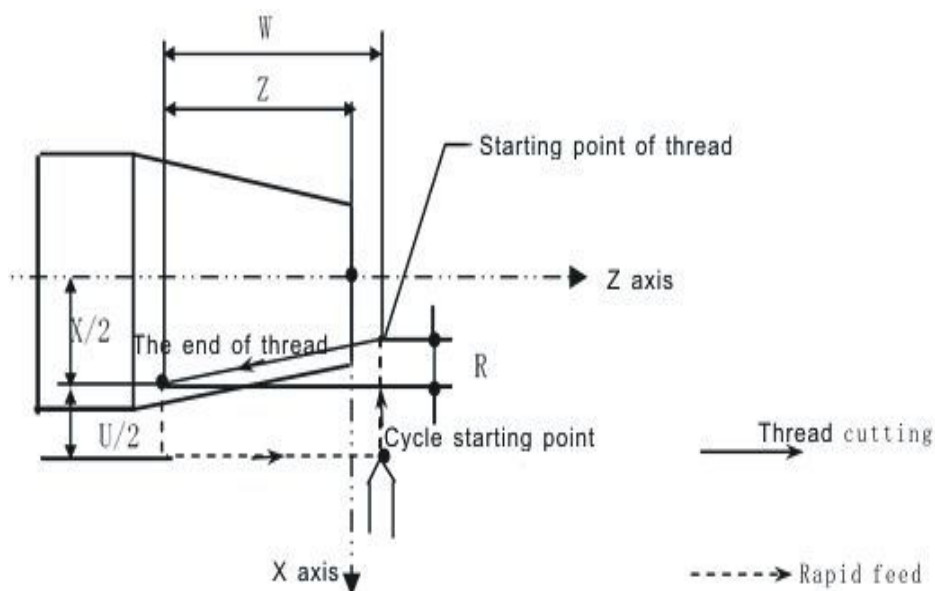
```

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___;

Cycle as shown in Fig, R is the X-axis direction in the thread cutting start point and the radius of the thread cutting end point difference. Cutting start point position cutting end point coordinate is less than a value of R in the negative X-axis direction, whereas R is positive.



Taper screw thread loop

For example:

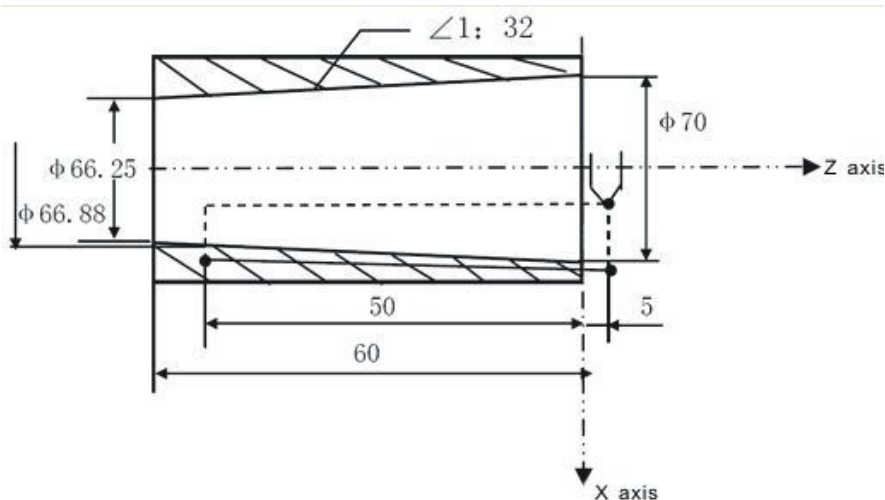
```

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 No.412 No.413;412 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); 413, 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 process 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_;

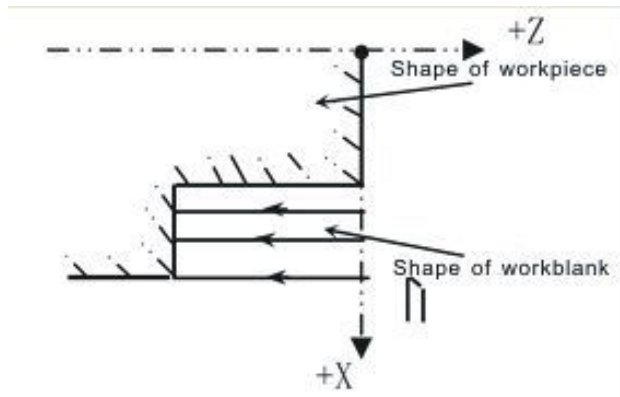
- J, K set the ratio of back tail X, Z. When J2 K1, X is twice faster than Z.
- P: back tail volume. Setting: 0.1 pitch. Set range: 1~255 (beyond to this range is invalid). The default value can be set by No.20 parameter in process parameter (Default when powering on).
- J, K, P are mode value.
- When 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.

(12) Usage for fixed cycle

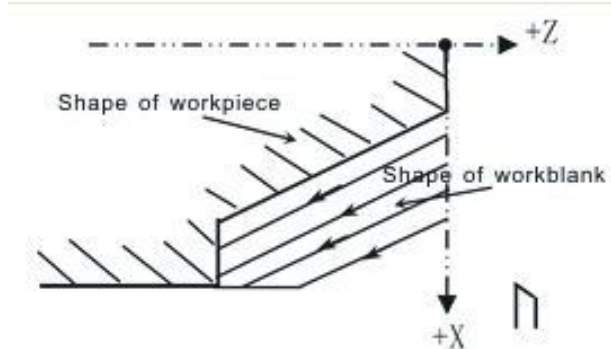
Could according to the shape of the workpiece and workblank to choose the suitable

fixed cycle.As follows:

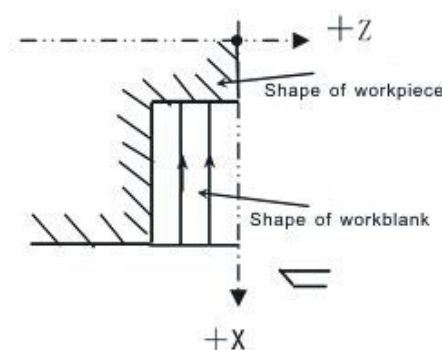
1) Cutting cylindrical cycle G90:



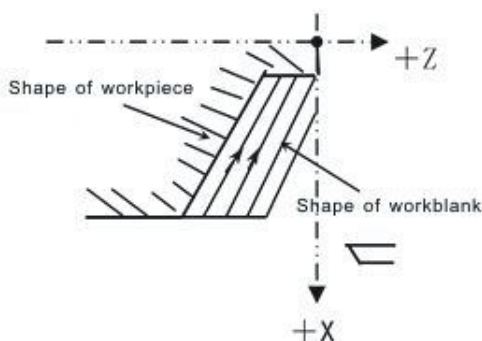
2) Taper cutting cycle G90:



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;
.....
```

(13) Fixed cycle of tapping (G93)

Format: 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..

example: G93 Z-100 F5 ; tapping cycle to Z-100;
 Z-101 ; tapping cycle to Z-101;
 G00 X50 ; G00

Tapping method of the X axis:

1) No.41 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.

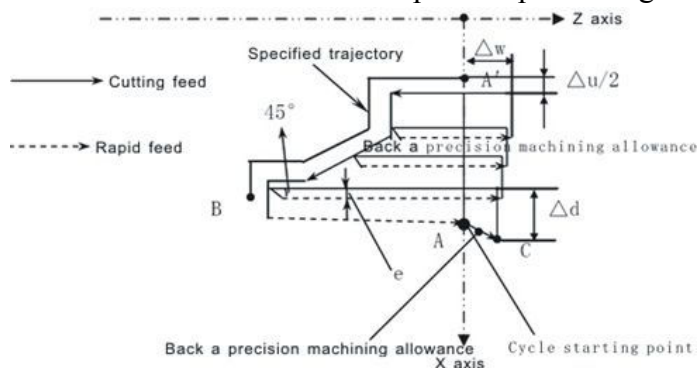
Note 6: When choosing the acceleration and deceleration control, if the spindle speed change, there is some delay when making the thread change. So choose the non-speed up or down if require the accuracy. However, with a step motor, the spindle speed can not be too high, otherwise cause the stuck.

(14) Column thick loop (G71)

Using G90, G92, G94, the program has been simplified to make some, but there is a class called Complex fixed cycle code (G70-G76), to make further simplify procedures, the use of these complex fixed cycle, just finishing instructions shape,

you can complete the whole process from roughing to finishing.

When given the shape of the road map process $A \rightarrow A' \rightarrow B$ and depth of cut Δd shown many times it will cut parallel to the Z-axis, then leave the final finishing lathe cutting margin Δw and $\Delta u / 2$ finished shape after processing.



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 signal; User parameter P1.

e: backward distance; User parameter P2.

ns: first N line.

nf: end N line.

Δu : X remain; User parameter P4.

Δw : Z remain; 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 $\rightarrow$ nf is effectively to fine machining, invalid in rough machining cycle.

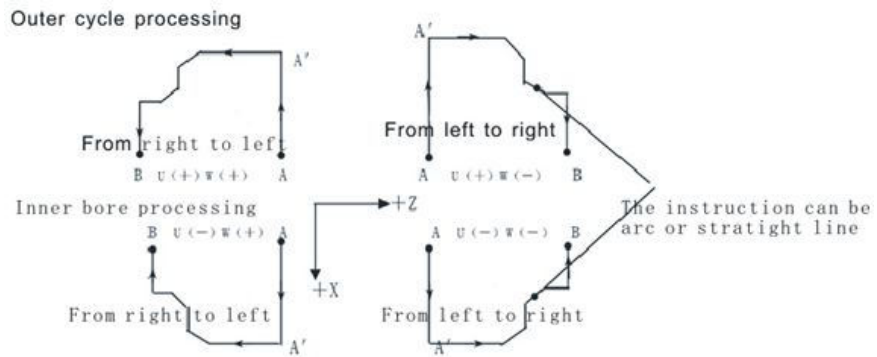
Note 2: A—B must conform the increase or decrease together mode of X and Z axis.

Note 3: G97 G96 is invalid for rough machining cycle when the program ns $\rightarrow$ 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 fine machining allowance, as shown below:



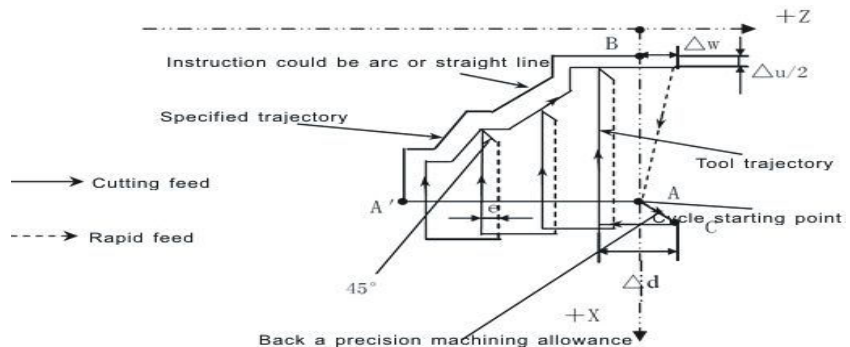
(15) End face thick loop(G72)

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) ;

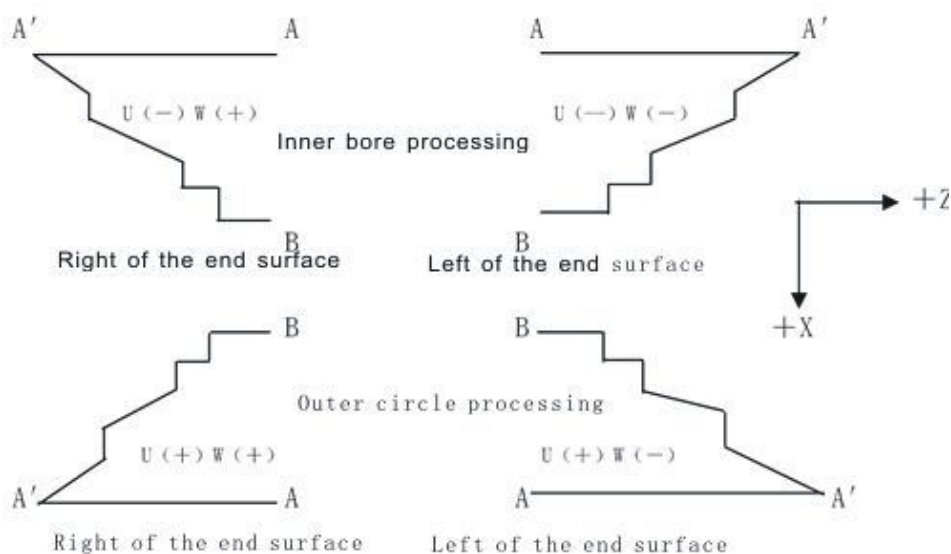
G72 P (ns) Q (nf) U (Δu) W (Δw) F (f) S (s) T (t) ;

Δd , e, ns, nf, Δu , Δw , f, s, t is same as G71.



Rough machining cycle of end surface

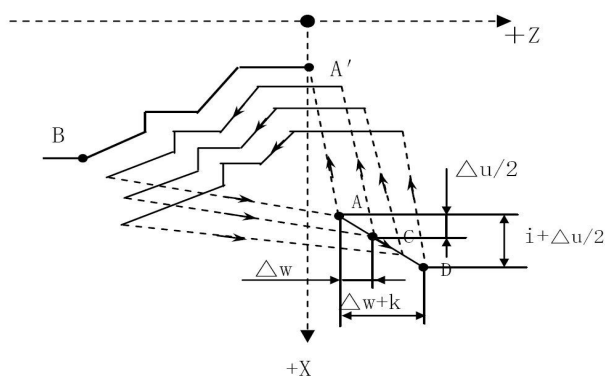
Use G72 to cut the shape, there are four situation. No matter what kind of is the tool parallel the X axis to cut again. Δu , Δw symbols are as follow



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.

(16) Close cutting loop(G73)

The so-called closed cutting cycle is to follow a certain cutting shape gradually approaching the final shape. This way for casting or forging blank cutting is a highly efficient way. As shown in Figure G73 cycle



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); --\

.....; |

. > A→A'→B, ns to nf

. |
. |
. |

N (nf); --/

i: X rough thickness; User parameter P7.

k: Z rough thickness; User parameter P8.

d: cutting times; User parameter P6.

Others is same as G71.

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.

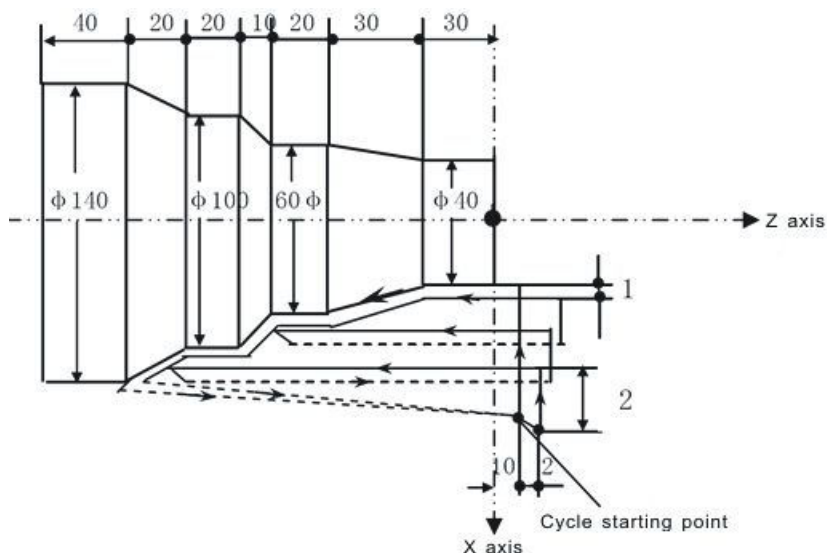
(17) Finish machining loop (G70)

Could use G70 to fine machining after using G71 G72 G73 to finish rough machining.

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.

For example1: G71 G70



N10 M03 S1500;

N20 T0101;

N30 G00 X160 Z10;

N40 G71 U2 R1;

N50 G71 P60 Q120 U2 W1 F100 S2000

N60 G00 X40;

N70 G01 Z-30 F80;

N80 X60 W-30;

N90 W-20;

N100 X100 W-10;

N110 W-20;

N120 X140 W-20;

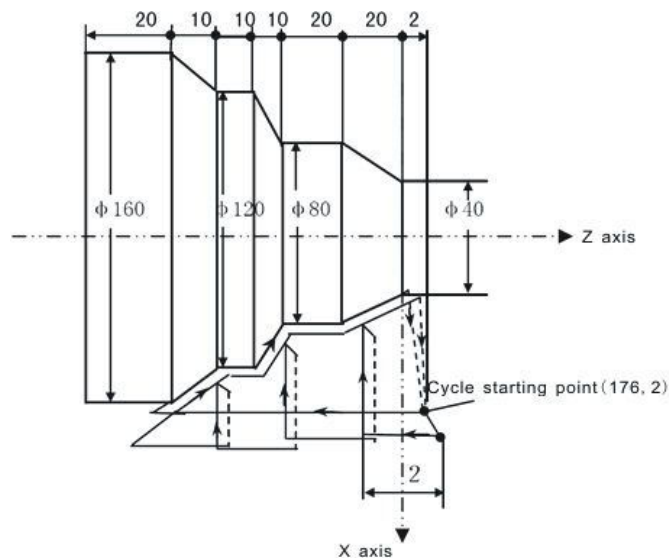
N130 G70 P60 Q120;

N140 G00 X200 Z50;

N150 T0100 M05;

N160 M30;

For example2: G72 G70



N10 M03 S2000;

N20 T0202;

N30 G00 X176 Z2;

N40 G72 W2 R1;

N50 G72 P60 Q120 U2 W1 F100 ;

N60 G00 Z-72;

N70 G01 X160 Z-70 F80;

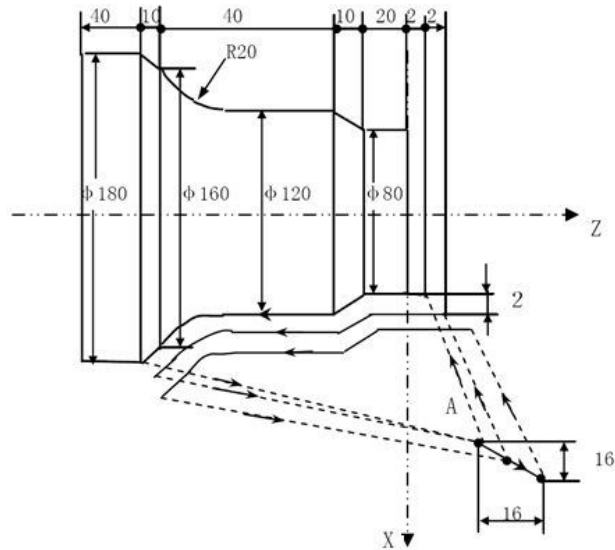
N80 X120 W10;

N90 W10;

N100 X80 W10;

N110 W20;

N120 X36 W22.08;
 N130 G70 P60 Q120;
 N140 G00 X200 Z50;
 N150 T0200 M05;
 N160 M30;
 For example3: G73 G70



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

(18) End face deep hole loop(G74)

Format:

G74 R (e) ;

G74 X (u) P (Δi) Z (w) Q (Δk) F (f) ;

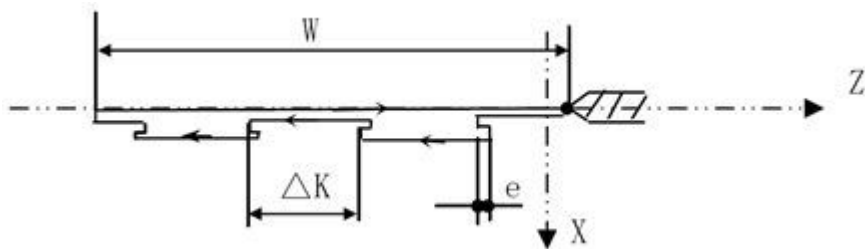
e: backward distance; User parameter P10.

Z (w) Z depth;

X (u) X end-point coordinate;

Δk : Z feed thickness; User parameter P9.

Δi : X feed thickness.



For example:

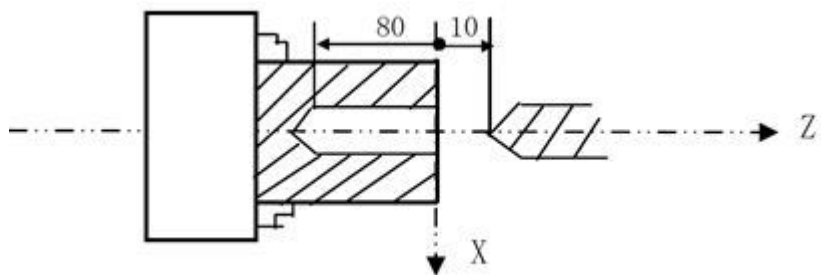
N10 G00 X0 Z10;

N20 G74 R2;

N30 G74 Z-80 Q10000 F800;

N40 G00 X50 Z50;

N50 M30;



(19) Slot cutting loop(G75)

Format:

G75 R(e)___;

G75 X(U)___ P(Δi)___ Z (w) ___ Q (Δk) ___ F(f)___;

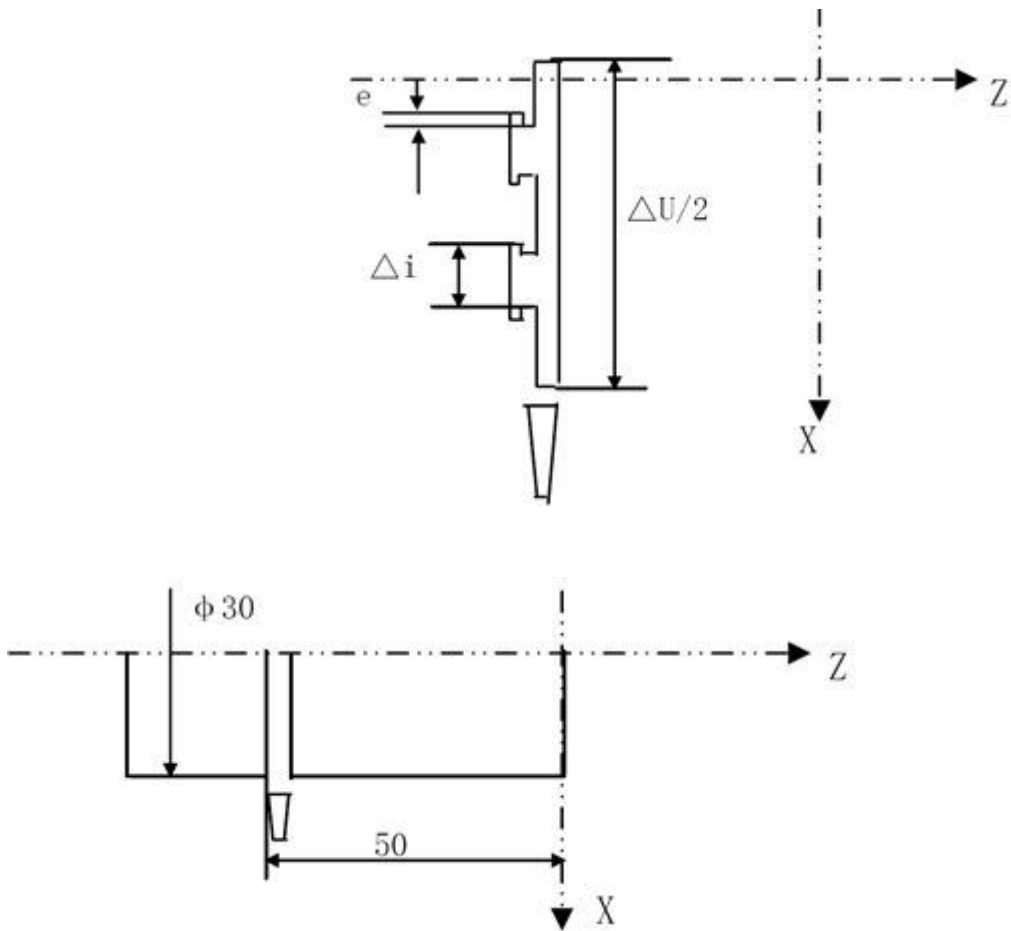
e: backward distance; User parameter P10.

X (u) X depth;

Z (w) Z end-point coordinate;

Δi : X feed thickness; User parameter P9.

Δk : Z feed thickness.



For example:

N10 M03 S1000;

N20 T0101;

N30 G00 X35 Z-50;

N40 G75 R1;

N50 G75 X-1 P5000 F60;

N60 G00 X100 Z50 M09;

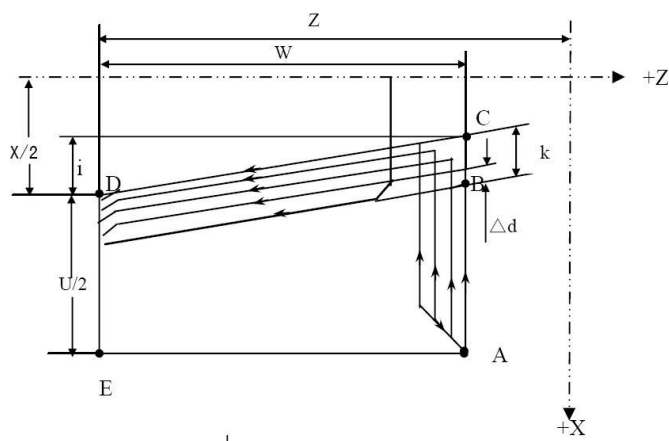
N70 M05;

N80 T0100;

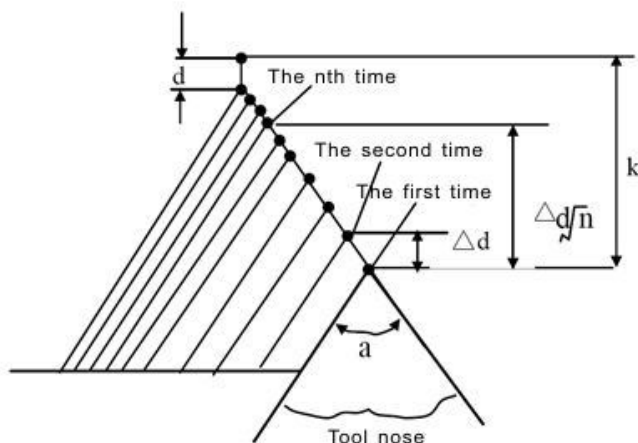
N90 M30;

(20) Complex screw thread loop(G76)

As fig:



Cutting method:



Format: G76 P(b)(c)(m)(r)(a) Q(Δ dmin) R(d) ;

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

b: 0——digression feed;

1——equidistance feed;

2——If the fist feed is too long in digression 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, User parameter P11.

r: quit length, User parameter P12.

a: thread tooth angle(degree),Usr parameter P13.

Δ_{min} : minimal cutting depth, User parameter P14.

d: finish turn remaining, User parameter P15.

I: X taper screw thread feed measure.

f: metric lead.

L: multiple thread head numbers.

SP: start angle: 0-360°

Δd : First cut amount (with G32 threading) in microns; or feed times. P24 G76 parameters specified by the user of the Q (Δd) meaning [P24 = 8 roughing infeed table number]. When this parameter P24 = 8, the address of the word Q (Δd) says many times as needed to complete the roughing cycle, the default is 1; otherwise address word Q (Δd) says that the first knife cutting depth, address word Q (Also there are equidistant feed and feed etc. regressive mode Δd) specify the number of times when feed.

All cases the amount of feed and feed roughing times are as follows:

1) $b = 0$, $P24 \neq 8$, every infeed depth: $\Delta d \sqrt{n}$;

2) $b = 0$, $P24 = 8$, each infeed depth is: the same way as a) according to Δd

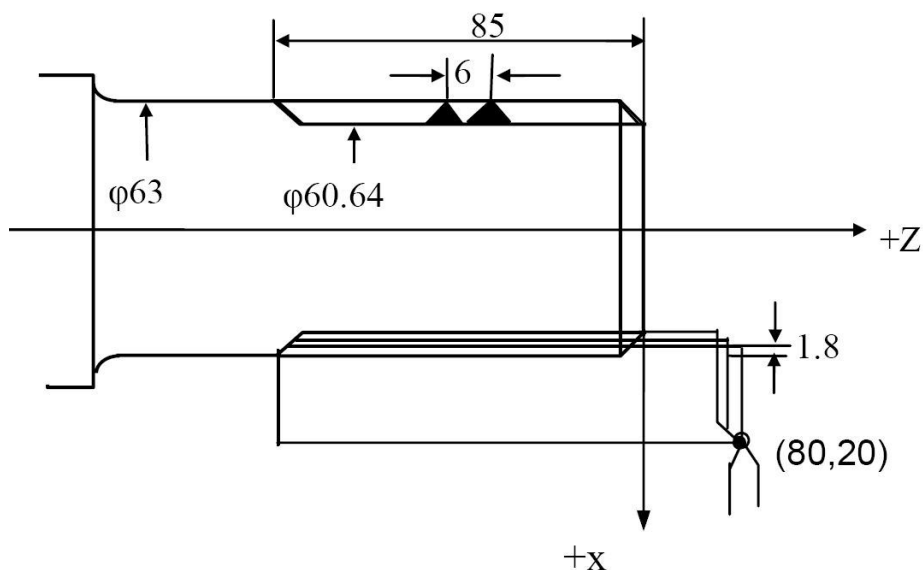
$$\frac{(K-d)}{\sqrt{\Delta d}} \sqrt{n}$$

calculated as: ,

Feed times for Δd ;

3) $b = 1$, $P24 \neq 8$, the amount of each feed: Δd , roughing feed number is $(kd) / \Delta d$;

4) $b = 1$, $P24 = 8$, the amount of feed for each: $(kd) / \Delta d$, roughing feed times for Δd ;



```

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.

(21) Fix loop(G22、 G800)

G22 is a program loop instruction, G800 is the end of the cycle instruction. Both must be paired for parts machining process requires repeated occasions. L is the number of cycles, ranging from 1-99999. Cycle instructions can be nested.

Format: G22 L-

```
.
.
.   G800 ;End
```

For example: N0000 M03 M08

N0001 G0 X200 Z200

N0002 G01 W-100 F300

N0003 G22 L6 ; loop 6 times

N0004 G01 U-22 F100

N0005 W-11 U6

N0006 W-30

N0007 W-10 U5

N0008 G0 U10

N0009 W51

N0010 G800 ; loop end

N0011 G26

N0012 M30

(22) Part coordinate setup(G52), Setup tool coor (G184, G185)

When programming in the workpiece coordinate system, to set sub coordinate system in order to programme.

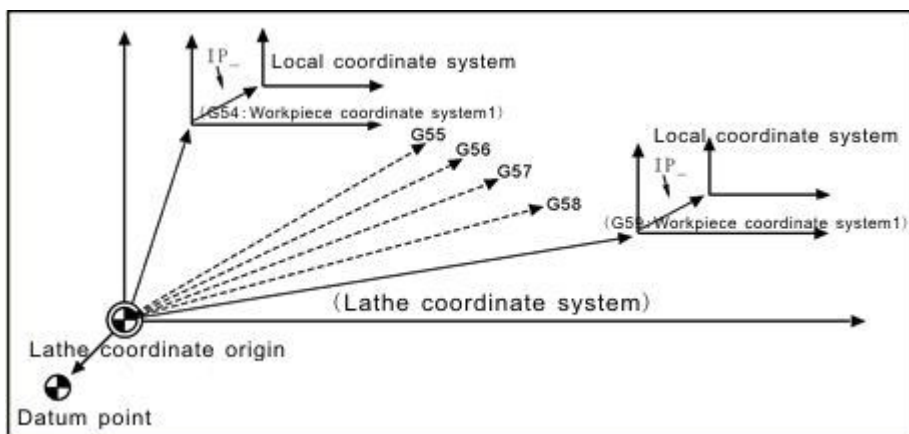
Format:

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.

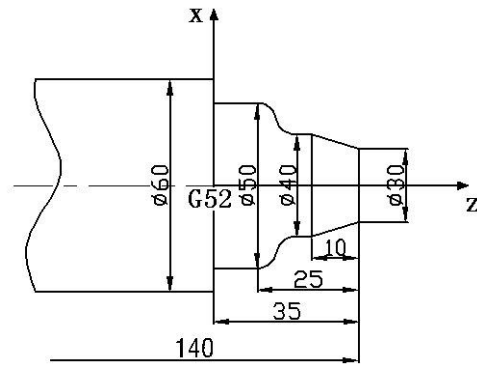
When setting local coordinate system, the motion with absolute mode is the coordinate value of local coordinate system. Use G52 to specify a new zero point which could change the position of local coordinate system. In order to cancel the local coordinate system and the specified coordinates in the workpiece 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
N3 T0101
N4 M03 S800 M08
N5 G01 Z35 F100
N6 X-1
N7 X70
N8 G71 U2 R1
N8 G71 P10 Q15 U0.5 W0.5 F150
N10 G01 X30
N11 X40 Z25
N12 Z20
N13 G02 X50 Z15 R5
N14 G03 X60 Z10 R5
N15 G01 Z0
N16 G00 X70
N17 G52 X0 Z0
N18 M05
N19 M30
    
```

For example: to achieve the same workpiece machining on a workpiece 5:

G22 L5

X50 Z60; workpiece

.....

G52 W-50; each Z-axis relative offset 50mm

G800

G52 X0 Z0; cancel the local coordinate system

G0 X100 Z100

M02

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

Back to the starting point of the program (the program first row starting N0000) instruction. G26 for the X, Z axes are starting to return, the same speed and G00.

Format:

G26、 G261、 G262、 G263、 G264、 G265; XZ、 X、 Y、 Z、 A、 B

Format:

N0000 G00 X120 Z300 ;

N0001 G01 X150 Z50 F160 ;

N0002 G26 ;

N0003 M2 ;

(24) Back to G25 point(G25、 G61、 G611/G615)

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, G615 returns B, G25 is the same as G26 if not set.

Format: G25 ;Save current coordinate

G61、 G611、 G612、 G613、 G614、 G615; XZ, X,Y,Z,A,B

For example:

N0000 G0 X20 Z80 ;

N0001 G01 U5 W-16 F200 ;

N0002 W-100 ;

N0003 G00 U10 ;

N0004 Z80 ;

N0005 G25

N0006 G01 U10 W-30 ;

N0007 G0 X100 Z200 ;

N0008 G61 ;

N0009 M2 ;

(25) continue feed cutting(G60/G64)

According to process requirements, you can pass G60 / G64 command specifies the connection between program segments.

Format: G60 ; cancel

G64 ; continue feed

(26) 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.

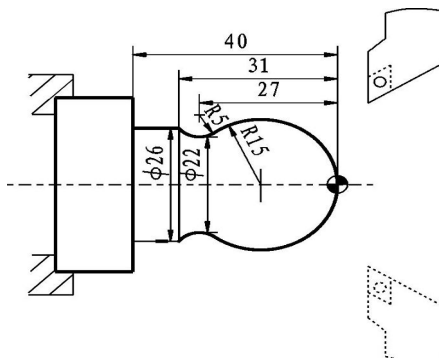
S:S value behind the G96 is the cutting constant linear speed, unit is m/min; S value behind the G97 is specifying spindle speed after cancel the constant linear speed; such as the default, is for the implementation of G96 Directive before spindle speed.

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
N2 M03 S400
N3 G96 S80
N4 G00 X0
N5 G01 Z0 F60
N6 G03 U24 W-24 R15
    
```

N7 G02 X26 Z-31 R5

N8 G01 Z-40

N9 X40 Z5

N10 G97 S300

N11 M30

(27) Feed mode(G98、 G99)

Format: G98 feed per minute instruction

G99 feed per revolution instruction

(28) Check skip(G31、 G311)

Specify G98 G01, G02, G03, etc sliced processing methods for feed per minute mode, with F designated feed rate. Range 0.01-15000mm / min.

Specify G99 G01, G02, G03, etc sliced processing methods for feed per revolution mode, specify the feed rate with F. Range 0.001-500mm / r.

When using the G99 command, the spindle encoder must be installed, otherwise it will be in a wait state. When the spindle speed is very slow (<10r / min) may be uneven process, please note that when used.

Format: G31 X_ Z_ F_ P_ ;No alarm

G311 X_ Z_ F_ P_ ;alarm

P:Nline+(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.

(29) Work coordinate(G53/G54/G55/G56/G57/G58/G59)

Used to select the workpiece coordinate system or machine coordinate system.

Format: G53(G54/G55/G56/G57/G58/G59)

G53 machine coordinate

G54 workpiece coordinate1

G55 workpiece coordinate 2

G56 workpiece coordinate 3

G57 workpiece coordinate 4

G58 workpiece coordinate 5

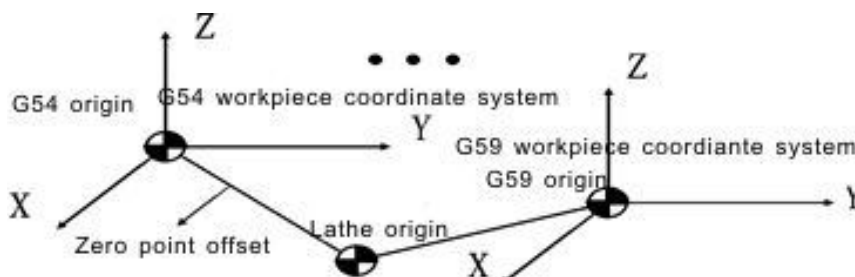
G59 workpiece coordinate 6

G53 machine coordinate system is defined by the machine first reference point (ie, machine zero) to determine the coordinate system. The first reference point coordinate in the machine coordinate system parameters can be set at the reference

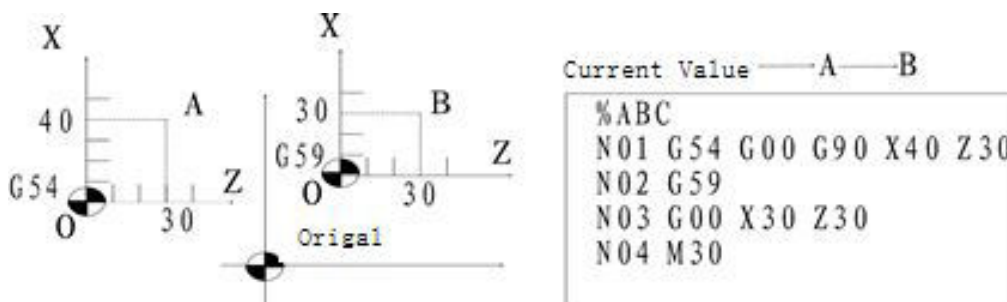
point, there should be a manual operation if the reference point coordinate values to modify the first reference point in the machine coordinate system later. G53 is the default coordinate system.

G54 / G55 / G56 / G57 / G58 / G59 workpiece coordinate system in the machine coordinate system offset by reference coordinate system

Number of setting.



For example:



Note: 1, before using this set of instructions, first set the coordinates of the origin of the coordinate values for each coordinate in the machine coordinate system.

2, before using this set of instructions, you must first reference point.

(30) Tool radius compensate G40、G41、G42

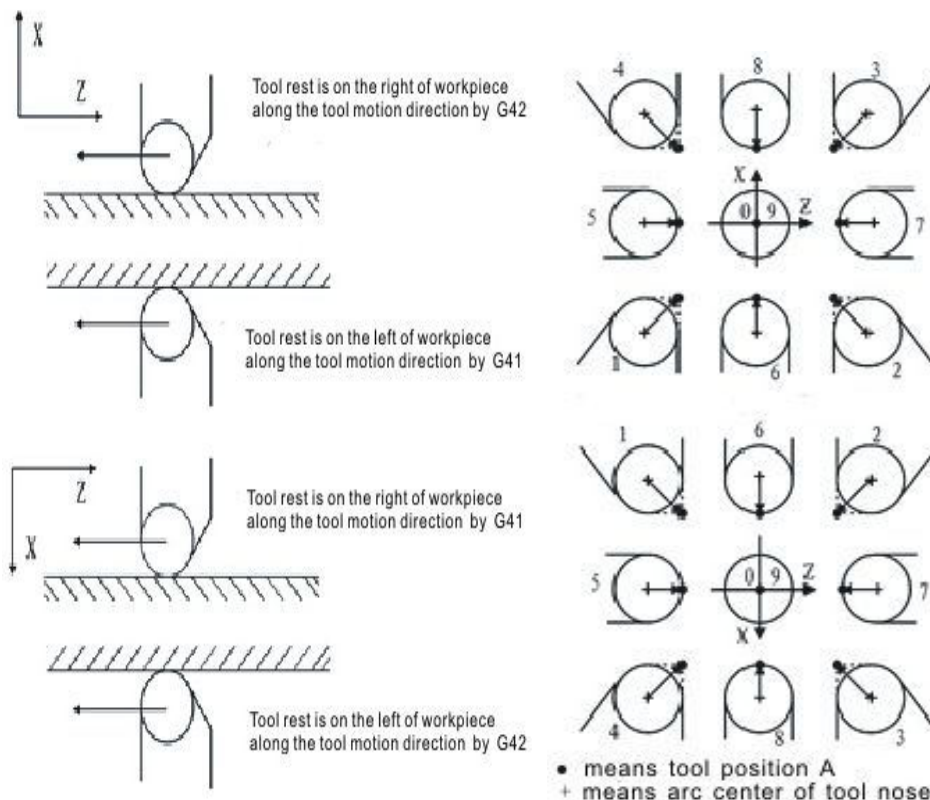
Format : G40 ; tool radius compensation cancellation

G41 ; tool radius compensation of cutter in the left of workpiece offset.

G42 ; tool radius compensation of cutter in the right of workpiece offset.

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.



Left and right compensate

Tool parameter

Note:

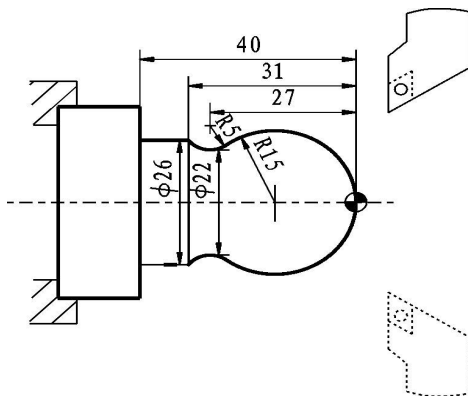
(1) G41 / G42 without parameters, the compensation number (on behalf of the tool tip radius compensation corresponding values) specified by the T code. Its tip arc offset number and tool offset number corresponding compensation. Establish and canceled .

(2) nose radius compensation with G00 or G01 instruction only, not the G02 or G03. Nose radius compensation tool compensation interface "radius compensation", the definition of the turning radius; imaginary tool nose number defines the direction of the tip.

The imaginary tool nose tip number defines the positional relationship between the tool and cutter tip arc center point, which is from 0-9 ten directions, as shown above.

Note: For the definition of the edge position code "imaginary tip" See section 3.8.4.

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
    
```

(31) 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:



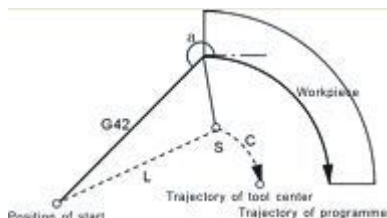
2) Tool motion when starting

The radius compensation without tool builds tool radius compensation

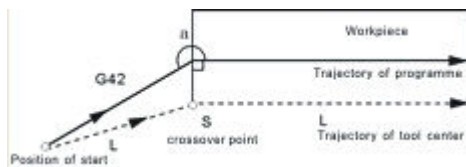
(a) Tool motion around the inside corner ($\alpha \geq 180$)

The tool center will move to the tool vector radius vertex of the starting point in next program line.

Straight line $\rightarrow$ Straight line



Straight line $\rightarrow$ Arc

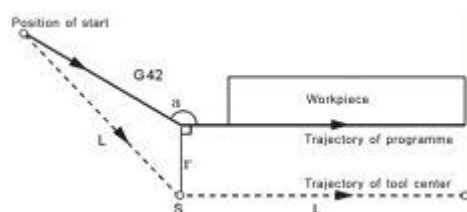


(b) The tool motion around the outside corner of obtuse angle ($90 \leq \alpha < 180$)

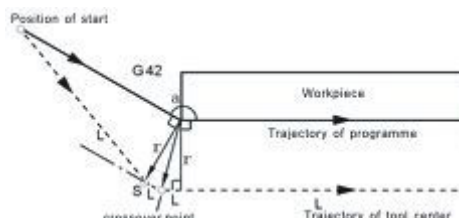
The tool center will move to the tool vector radius vertex of the end point in this program line.

Straight line $\rightarrow$ Straight line

A type

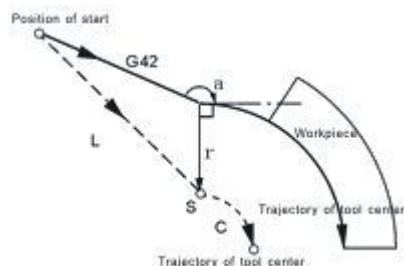


B type

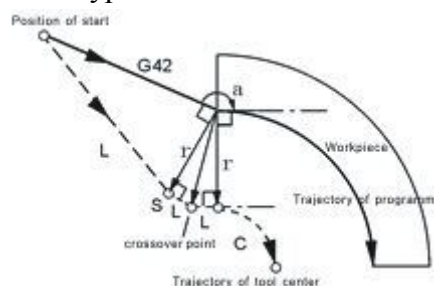


Straight line $\rightarrow$ Arc

A type



B type



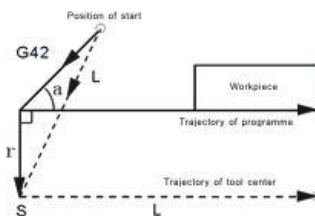
(c) The tool motion around the outside corner of acute angle ($\alpha < 90$)

The tool center will move to the tool vector radius vertex of the end point in this program line.

Straight line $\rightarrow$ Straight line

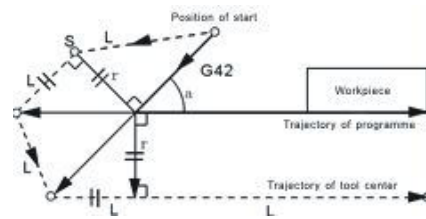
A type

B type

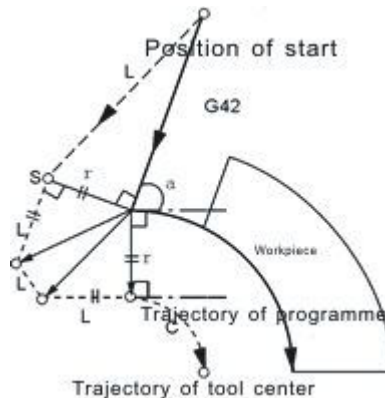
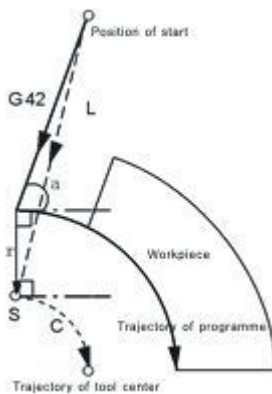


Straight line->Arc

A type



B type

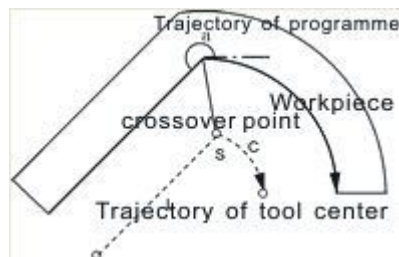
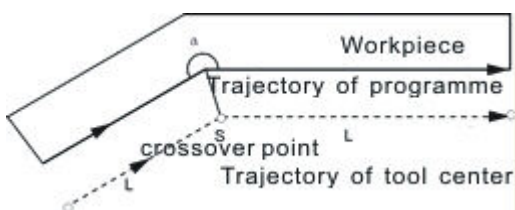


3) Tool motion in offset mode

(a) Tool motion around the inside corner ($180 \leq \alpha$)

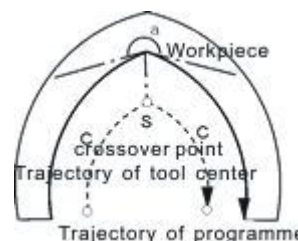
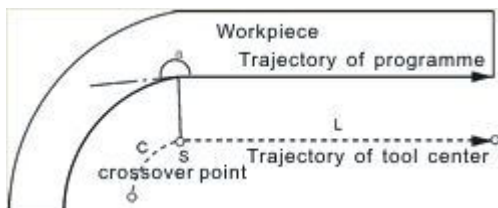
Straight line->Straight line

Straight line->Arc



Arc-> Straight line

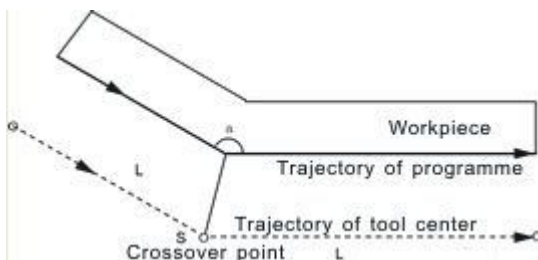
Arc->Arc



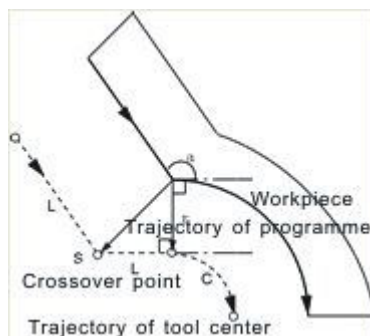
(b) The tool motion around the outside corner of obtuse angle ($90 \leq \alpha < 180$)

Straight line ->Straight line

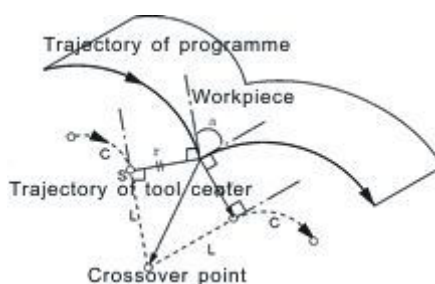
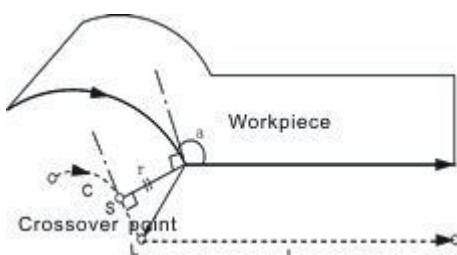
Straight line ->Arc



Arc-> Straight line



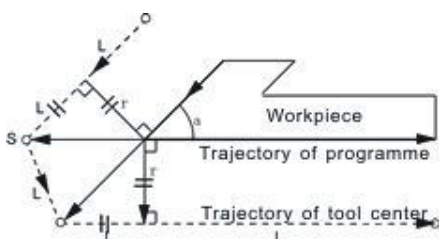
Arc->Arc



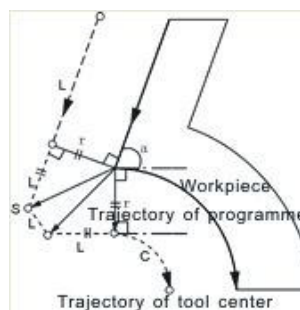
(c) The tool motion around the outside corner of acute angle ($\alpha < 90^\circ$)

Straight line->Straight line

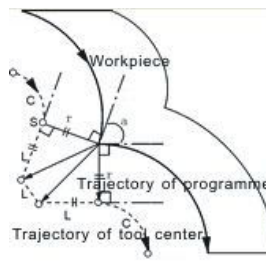
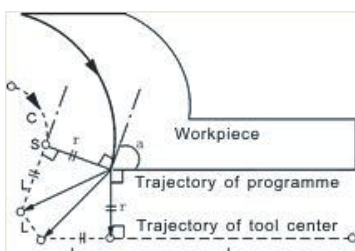
Straight line->Arc



Arc-> Straight line



Arc->Arc

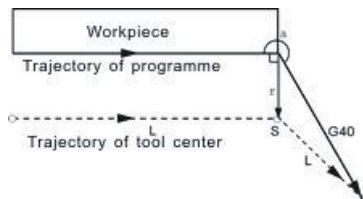


4) Tool motion in offset-cancel mode

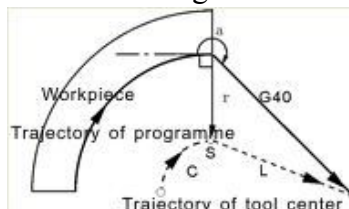
(a) Tool motion around the inside corner ($180 \leq \alpha$)

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

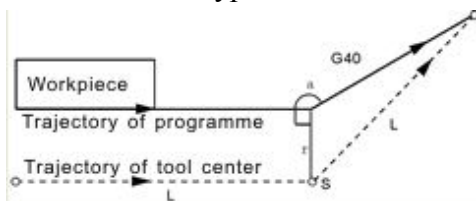


(b) The tool motion around the outside corner of obtuse angle ($90^\circ \leq \alpha < 180^\circ$)

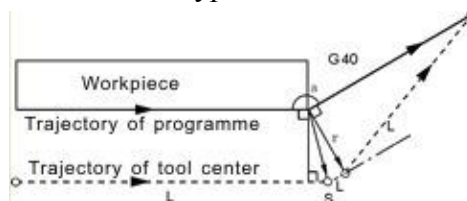
The tool center will move to the tool vector radius vertex of the starting point in next program line.

Straight line → Straight line

A type

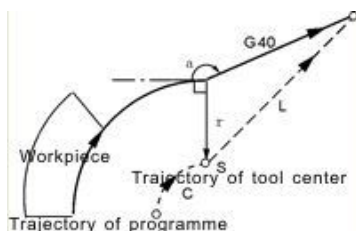


B type

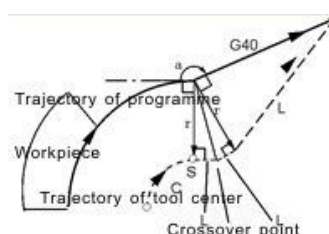


Arc → Straight line

A type



B type

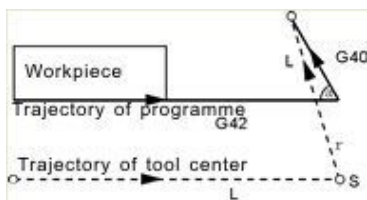


(c) The tool motion around the outside corner of acute angle ($\alpha < 90^\circ$)

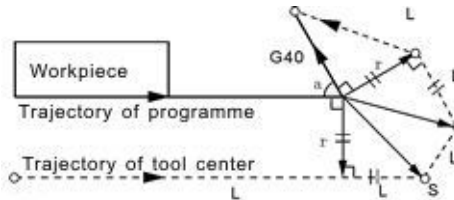
The tool center will move to the tool vector radius vertex of the starting point in next program line.

Straight line → Straight line

A type



B type

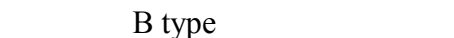


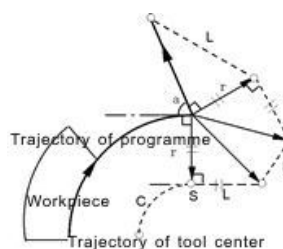
Arc → Straight line

A type



B type





(32) Automatic beveling (I) and smoothing(R)

Format :

G01(G00) X I automatic beveling, the coordinate in the next program segment must be G01(G00) Y.

G01(G00) Y I automatic beveling, the coordinate in the next program segment must be G01(G00) X.

G01(G00) X R automatic smoothing, the coordinate in the next program segment must be G01(G00) Y.

G01(G00) Y R automatic smoothing, the coordinate in the next program segment must be G01(G00) X.

Format for G18:

G01(G00) X I automatic beveling, the coordinate in the next program segment must be G01(G00) Z.

G01(G00) Z I automatic beveling, the coordinate in the next program segment must be G01(G00) X.

G01(G00) X R automatic smoothing, the coordinate in the next program segment must be G01(G00) Z.

G01(G00) Z R automatic smoothing, the coordinate in the next program segment must be G01(G00) X.

Format for G19:

G01(G00) Y I automatic beveling, the coordinate in the next program segment must be G01(G00) Z.

G01(G00) Z I automatic beveling, the coordinate in the next program segment must be G01(G00) Y.

G01(G00) Y R automatic smoothing, the coordinate in the next program segment must be G01(G00) Z.

G01(G00) Z R automatic smoothing, the coordinate in the next program segment must be G01(G00) Y.

Pay attention:

1. The address of I and R are specified with radius model. The running distance of this line and the next line must be greater than the length of beveling or radius of smoothing, otherwise the system will decrease the length of beveling or radius of smoothing to minimal running distance of this line and the next line automatically.

2. The two adjacent lines must be 90 degrees.

For example:

```

0 G54 G0 X-50 Y-50 Z20
N1 M03 S500
N2 G01 G42 D01 X0 Y0 F200
N3 G01 Z-5
N4 X100 I4           ; Beveling4x4
N5 Y40 R6           ; SmoothingR6
N6 X47 R5           ; SmoothingR5
N7 Y70 I3           ; Beveling3x3
N8 X15
N9 X0 Y40
N10 Y0
N11 G0 X-50 Y-50 G40
N12 Z50
N13 M30
    
```

(33) Pole coordinate program(G15/G16)

Polar coordinates input directive allows radius and angle in polar coordinates, the angle of the positive Z direction is counterclockwise turned, while the negative direction is a clockwise turn. Radius with absolute command value instruction (Z), the angle with absolute command (X).

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

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

Description:

- 1, Z specify zero as the origin of the polar coordinate system from the point of measuring the radius of the workpiece coordinate system.
- 2, IP- the polar coordinate system selection plane axis address their values:
 Z axis: polar coordinate radius
 X-axis: Polar angle
- 3, Z set to zero as the origin of the polar coordinate system workpiece coordinate system:

By absolute programming instructions a specified radius (distance between the zero point and programming). Zero set workpiece coordinate system for the origin of the polar coordinate system

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

Metric conversion instructions select English or Metric input.

Format:

G20 ; inch;
G21 ; millimeter;

Description: The G code must be compiled at the beginning of the program, before setting the coordinate system to specify a separate block. G code designated for inch / metric conversion of input data after switching to the smallest unit of imperial or metric input increment angle data input unit remains unchanged, changes in the value of the following units after inch / metric conversion:

- by F code feed rate
- position command
- workpiece zero point offset value
- tool offsets
- scale unit manual pulse generator
- in incremental feed of the moving distance

(35) Call sub-program (M97/M98/M99)

Unconditional jump

Line running jump conditions specified by P M97 P; P4 said entrance line number with 4 field numbers specified program transfer main program (mark line).

Subroutine call

In this system the subroutine should be an independent program.

M98 P L unconditionally subroutine call instruction. P subroutine path and name specified procedure call, L refers to a subroutine call number address.

The M98 instruction can be omitted without writing, format: PP file name, the file name can be hidden files, hidden files first character must be a "HIDEFILE" at the beginning. If the file "HIDEFILE01", the program in the program area is not displayed, you can use command M98 PHIDEFILE01 or M98 P*01 or PP*01 or PPHIDEFILE01 call.

For example: Psub/1390 table subroutine tmp/NC/sub/1390

Note: 1, tmp/NC/ for the system program the default path, sub a folder for the following

2, the subroutine must be a separate program.

Methods the subroutine call in 3, USB USB: P[or P].

For example: M98 P[A1234 says call subroutines in A1234 usb;

M98 P]SS12 says call subroutines in SS12 usb;

PP[FFDE says call subroutines in FFDE usb;

If the folder called USB in the subroutine, the path needs to be compiled on the file.

Subroutine call number address character before L must have spaces. Subroutine is run to the end, return to the next section of the program of the main program. If a program contains a fixed sequence or repeated pattern of words, this order or graphics can be compiled subroutine in memory storage to simplify programming subroutine by the main program calls the subroutine is also calls another subroutine.

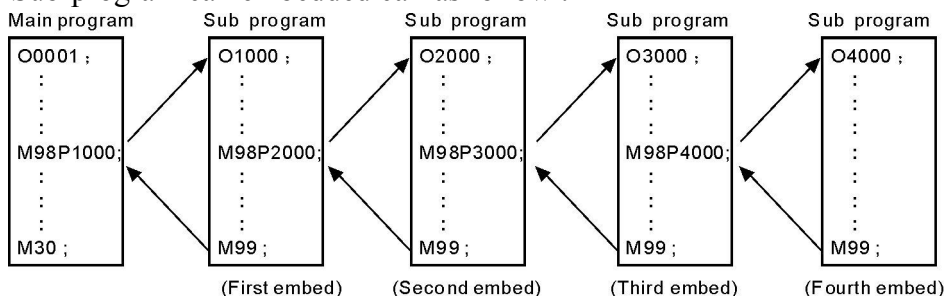
M99 end of subroutine return instruction, end of subroutine must have this instruction.

Note:

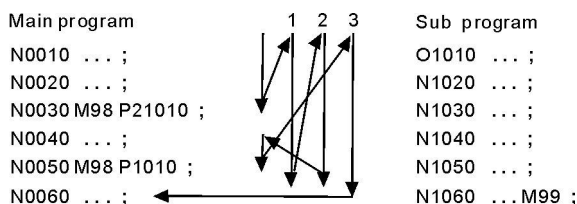
- 1) the M99 instruction in the main program with M02;
- 2) M99 with P in the main program with the M97 command;
- 3) M99 the main program calls back at downlink in the subroutine;
- 4) M99 with P in the subroutine return main program P program for;

Subroutine call other subroutines (nested). Whether the application is the main program or subroutine, subroutine return is returned to the application for the next program for. When the main program subroutine calls, it is considered a level subroutine. Calls can be nested class 4 call see chart.

The Sub-program can embedded call as follow :



For example: In the same way subroutine call the main program subroutine calls another subroutine



The calling instruction can be used for 9999 times in the most.

(36) Conditional wait, jump instruction

The system of M code is used for detecting the external input signal as the condition, as follows:

Conditions wait

M12 M13 instruction are used to detect the input signal M12, M12 in program line is to detect M12 input signal is effective to execute the next program line, M13 means to detect M12 input signal is invalid to execute the next program line. The instruction is in an independent line.

M14 M15 instruction are used to detect the input signal M14, M14 in program line is to detect M14 input signal is effective to execute the next program line, M15 means to detect M14 input signal is invalid to execute the next program line. The instruction is in an independent line.

M16 M17 instruction are used to detect the input signal M16, M16 in program line is to detect M16 input signal is effective to execute the next program line, M17 means to detect M16 input signal is invalid to execute the next program line. The instruction is in an independent line.

M18 M19 instruction are used to detect the input signal M18, M18 in program line is to detect M18 input signal is effective to execute the next program line, M19 means to detect M18 input signal is invalid to execute the next program line. The instruction is in an independent line.

M22 M23 instruction are used to detect the input signal M22, M22 in program line is to detect M22 input signal is effective to execute the next line program, M23 means to detect M22 input signal is invalid to execute the next program line. The instruction is in an independent line.

M28 M29 instruction are used to detect the input signal M28, M28 in program line is to detect M28 input signal is effective to execute the next line program, M29 means to detect M28 input signal is invalid to execute the next program line. The instruction is in an independent line.

Conditional jump

Plus Pxxxx (number of program line) in front of the M12 /M13 /M14 /M15 /M16 /M17 /M22 /M23 /M28 /M29 instruction. Shifting if the condition success, otherwise execute the next.

For example: M14 P0120

When the program running to this line and the system detecting the M14 input signal effectively, program will jump to the 120th line of program (the marking line), execute the next instruction if the M14 input signal is invalid.

(37) T tool

Format function

Tab a: Tool number, b: compensate number

Format:

N0000 T0101

N0001 G0 X30 Z500

N0002 T0303

N0003 G00 X50

N0004 T0505

N0005 M02

(38) S、SS SP speed

It is the mode, actual running speed is the setting speed times the trimming rate of speed,

F is used for specify the processing speed of feeding instruction G01 G02 G03.

The range is 0.01-15000mm/min, feeding speed is Fx trimming speed, F has mode function.

Executing the F instruction at the first, and then execute the motion instruction when the F instruction and motion instruction are in the same line.

(39) F feed speed

The system offers two ways spindle controlling modes.

The first spindle speed is specified by S, the first spindle has two kinds of gear controlling mode:

(1) The first is four gear spindle speed electrical control, output four bits code of step speed change, M41-M44 instruction control corresponds to S01-S04 output code, step speed change. Use No.50 No.51 No.52 No.53 and No.54 parameter in axis parameter to set the mode of shifting.

(2) The second uses four gears + step-less speed, M41-M44 instruction control, correspond the output S01-S04 code. Use No.42 No.43 No.44 No.45 parameter in speed parameter to set the maximum speed of corresponding gear, use No.50 No.51 No.52 No.53 No.54 parameter in axis parameter to set the mode of shifting.

Stepless speed, the range is 0-99999, output 0-10V variable-frequency voltage. The output voltage trims x10V of maximum speed of specified spindle.

Second spindle speed is specified by SS, the highest speed is controlled by the No.46 parameter in speed parameter, output 0-10V variable-frequency voltage.

(40) 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.

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

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 GOO 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

```

(41) 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)

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

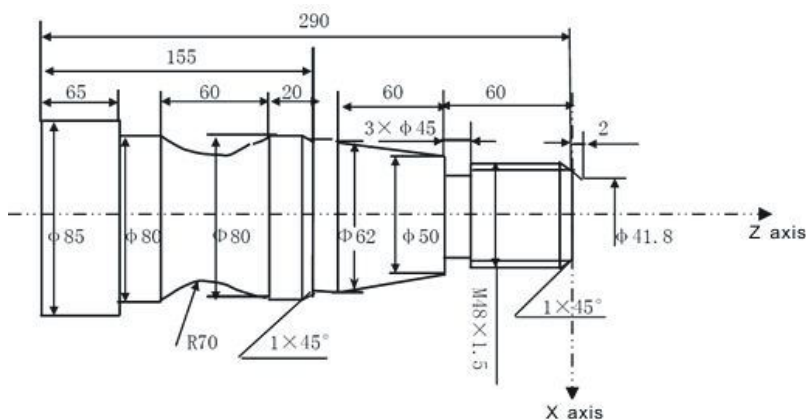
```

4.4 Synthetic instance for programming

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 3m; 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; Chamber
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 ex-circle
N90 X78;            Tool is backing
N100 X80 W-1;       Chamber
N110 W-19;          Fine machining in  $\phi 80$ mm ex-circle
N120 G02 X80 W-60 R70; Fine machining in arc (I63.25 K-30)

```

N130 G01 Z-225; Fine machining in $\Phi 80$ mm ex-circle
 N140 X85; Tool is backing
 N150 Z-290; Fine machining in $\Phi 85$ mm ex-circle
 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 $\Phi 45$ mm 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

4.5 Usage for grinder

Use for ex-circle 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、Ex-circle grinding mode T0101

M61 ; Use M61 to choose the mode of ex-circle 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 ; Ex-circle rough grinding in face enough M22, make sure within 3mm

G31 U-2 F10 P1026 ; Ex-circle fine grinding in face enough M24, make sure within 2mm

G31 U-1 F1 P1025 ; Ex-circle 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

G0 U-0.01 ; Fix 0.05mm

G01 W-20 F10

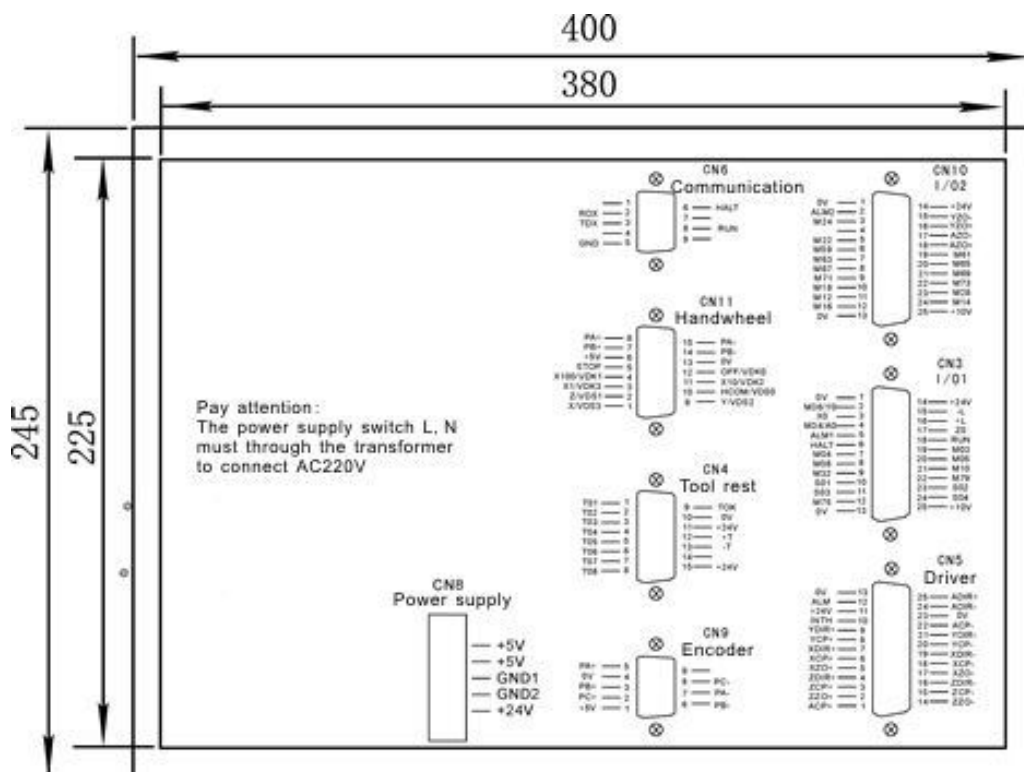
W20

G800

G185 X100

M02

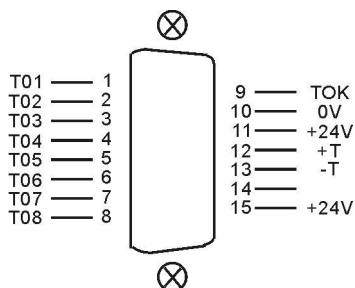
5.3 system rear view



Note: switching power supply L, N must be connected to AC 220V, current 0.5A through isolation transformer.

5.4 interface connection graph

5.4.1 CN4 and electric tool carrier connection

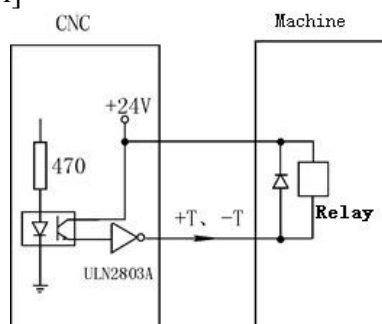


CN4 DB15(hole) electric tool carrier				
signal	pin	I/O	function	availability
0V	10	OUT	0V	0V
+24V	11、15	OUT	+24V	+24V
+T	12	OUT	Positive rotate	0V

-T	13	OUT	Reverse rotate	0V
T1	1	IN	T1 signal	0V
T2	2	IN	T2 signal	0V
T3	3	IN	T3 signal	0V
T4	4	IN	T4 signal	0V
T5	5	IN	T5 signal	0V
T6	6	IN	T6 signal	0V
T7	7	IN	T7 signal	0V
T8	8	IN	T8 signal	0V
TOK	9	IN	Lock signal	0V

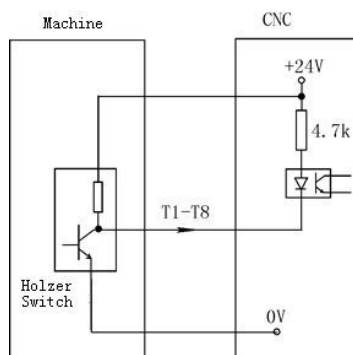
System can control 1-99 Tools. Interfix Tool parameter as:

- 1,Active tool function
[1 mean Yes,0 mean No]
- 2,Active tool number
- 4,Tool positive rotate max-time(s)
- 5,Delay time after tool positive rotate(ms)
- 6,Delay time after tool stop(ms)
- 7,Tighten time of tool reverse rotate(ms)
- 9,Have total signal TOK(1 mean have)
- 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,Active tool mode
[1 mean normal,0 mean coding tool]

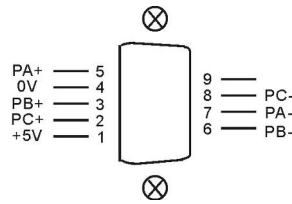


System output signal +T、-T:

Tool input T1~T8、TOK signal:

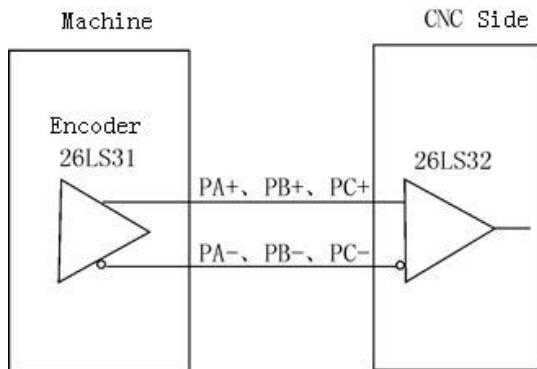


5.4.2 CN9 and spindle encoder connection

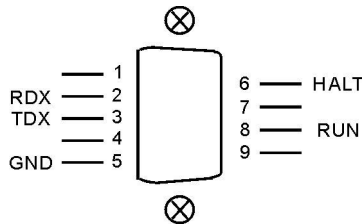


CN9 DB9(pin) spindle encoder				
signal	pin	I/O	function	availability
0V	4	OUT	0V	0V
+5V	1	OUT	+5V	+5V
PA+	5	IN	+A signal	5V
PA-	7	IN	-A signal	
PB+	3	IN	+B signal	5V
PB-	6	IN	-B signal	
PC+	2	IN	+Z signal	5V
PC-	8	IN	-Z signal	

Encode input signal PA、PB、PC:



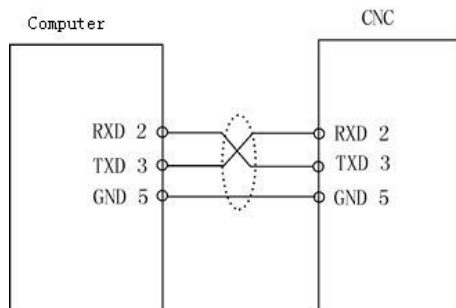
5.4.3 CN6 and computer system connection



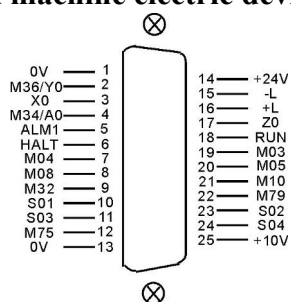
CN6 DB9(hole) RS232 communication				
signal	pin	I/O	function	Availability
0V	5	OUT	0V	0V
RXD	2	IN	RXD	
TXD	3	OUT	TXD	

RUN	8	IN	run	0V
HALT	6	IN	pause	0V

CN6 connect fig:



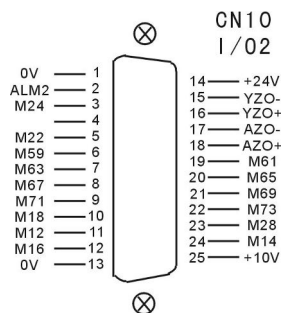
5.4.4 CN3 and machine electric device I/O1 connection



CN3 DB25(hole) I/O1 machine signal				
signal	pin	I/O	function	Availability
0V	1	OUT	0V	0V
+24V	14	OUT	+24V	+24V
M36/Y0	2	IN	M36/Y0	0V
X0	3	IN	X axis Zero	0V
Z0	17	IN	Z axis Zero	0V
-L	15	IN	Positive limit	0V
+L	16	IN	Negative limit	0V
M34/A0	4	IN	M34/A0	0V
ALM1	5	IN	Transducer alarm l	0V
HALT	6	IN	Pause	0V
RUN	18	IN	Run	0V
M03	19	OUT	spindle clockwise	0V
M04	7	OUT	SP counter clockwise	0V
M05	20	OUT	SP stop	0V
M08	8	OUT	coolant	0V
M10	21	OUT	spindle chuck	0V
M32	9	OUT	lubricating	0V
M79	22	OUT	spindle tailstock	0V

S01	10	OUT	spindle first gear	0V
S02	23	OUT	spindle second gear	0V
S03	11	OUT	spindle third gear	0V
S04	24	OUT	spindle fourth gear	0V
M75	12	OUT	C axis mode	0V
+10V	25	OUT	the first spindle converting	0~10V
0V	13	OUT	0V	0V

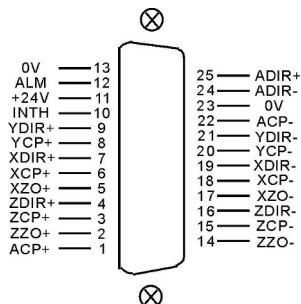
5.4.5 CN10 and machine electric device I/O2 connection



CN10 DB25(hole) I/O2 machine signal				
signal	pin	I/O	function	availability
0V	1	OUT	0V	0V
+24V	14	OUT	+24V	+24V
ALM2	2	IN	Machine alarm2	0V
M24	3	IN	M24	0V
M22	5	IN	M01 input	0V
M59	6	OUT	Huff	0V
M61	19	OUT	M61	0V
M63	7	OUT	M63	0V
M65	20	OUT	M65	0V
M67	8	OUT	M67	0V
M69	21	OUT	M69	0V
M71	9	OUT	M71	0V
M73	22	OUT	M73	0V
M18	10	IN	M18	0V
M28	23	IN	M28	0V
M12	11	IN	M12	0V
M14	24	IN	M14	0V
M16	12	IN	M16	0V
YZO+	16	IN	+Ymotor Zero signal	5V
YZO-	15	IN	-Ymotor Zero signal	
AZO+	18	IN	+Amotor Zero signal	5V
AZO-	17	IN	-Amotor Zero signal	

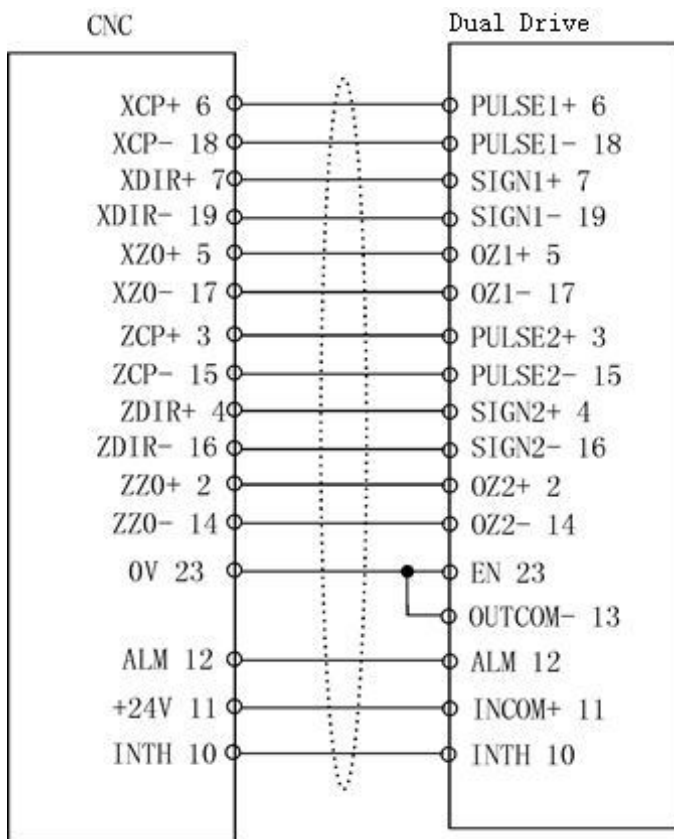
+10V	25	OUT	the second spindle converting	0~10V
0V	13	OUT	0V	0V

5.4.6 CN5 and servo drive & motor connection

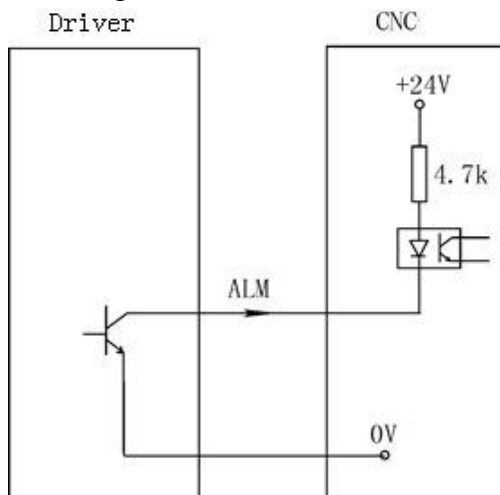


CN5 DB25(pin) servo drive signal				
signal	pin	I/O	Function	Availability
XCP+	6	OUT	X pulse signal +	5V
XCP-	18	OUT	X pulse signal -	
XDIR+	7	OUT	X direction signal +	5V
XDIR-	19	OUT	X direction signal -	
YCP+	8	OUT	Y pulse signal +	5V
YCP-	20	OUT	Y pulse signal -	
YDIR+	9	OUT	Y direction signal +	5V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

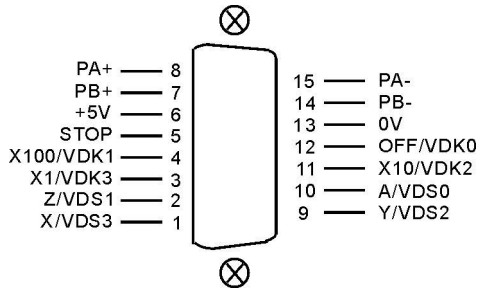
CN5 X、Z connect to our Co.'S servo drive:



Servo alarm signal:



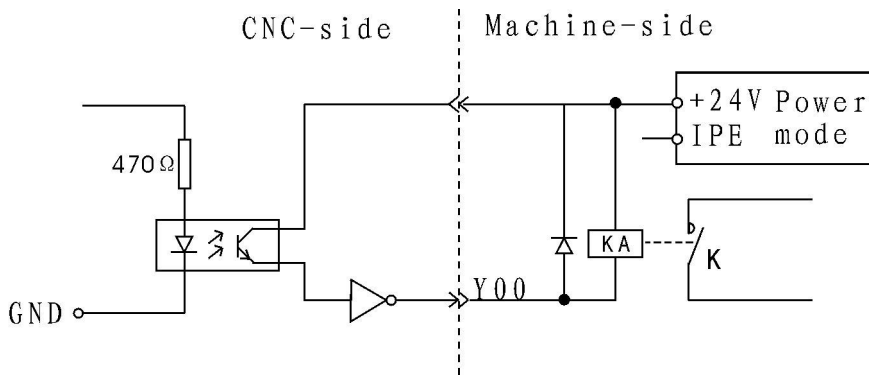
5.4.7 CN11 and hand wheel, band switch connection



CN11 DB15(pin) hand wheel, band switch connection				
signal	pin	I/O	function	Availability
0V	13	OUT	0V	0V
+5V	6	OUT	+5V	+5V
PA+	8	IN	A signal +	5V
PA-	15	IN	A signal -	
PB+	7	IN	B signal +	5V
PB-	14	IN	B signal -	
STOP	5	IN	emergency stop	0V
OFF/VDK0	12	IN	Off/ feed amending 0	0V
X100/VDK1	4	IN	*100/ feed amending 1	0V
X10/VDK2	11	IN	*10/ feed amending 2	0V
X1/VDK3	3	IN	*1/ feed amending 3	0V
A/VDS0/HALT	10	IN	A/SP amending 0/halt stop	0V
Z/VDS1	2	IN	Z/SP amending 1	0V
Y/VDS2/RUN	9	IN	Y/SP amending 2/run	0V
X/VDS3	1	IN	X/SP amending 3	0V

5.4.8 General, motion control I/O output port principle which is availability by "0V"

Output port of Y00-Y23 are availability by "0V", the connection method as follow (take Y00 control relay as example):

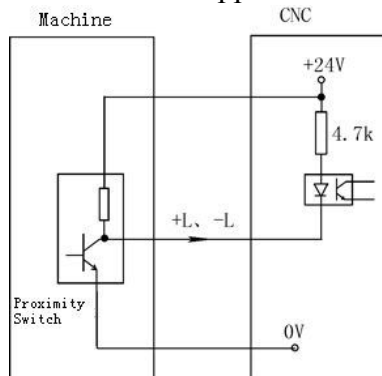


Special Cautious: Because the output ports are the transistor output, thus the load electric current cannot be bigger than 150mA.

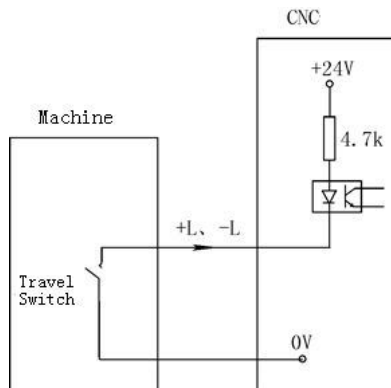
5.4.9 Reference points connections input port principle

(take +L、-L axis as example)

Model1: NPN approach switch



Mode2: general switch



Chapter 6 System's daily maintenance and repair

In order to plenty use CNC system's function and promote efficiency, the most important work is correctly using system, and notice system's daily maintenance work, promote Mean Time Between Failures MTBF. Now this system's maintenance method is introduced as follows:

6.1 System's maintain

6.1.1 System's using must be under the good circumstance.

6.1.2 Operator、programmer and repairer must be familiar with NC machining technology, and according the require of user book correctly use, do one's best to avoid improper operation.

6.1.3 Everyday operator should clean the system's box and panel in case for corrupt thing and sundries to damnify it.

6.1.3 When CNC system's using time is over three month, operator should open the system box and clean inside.

6.1.4 If not using system for long time, should boot the system one time every week.

6.2 Ordinary trouble

6.2.1 System can't boot

- 1) check if power is normal.
- 2) check if power switch is turn on.
- 3) check insurance.

6.2.2 No display as boot

- 1) Boot again or reset.
- 2) Check if switch power's +5V、+12V、-12V、-24V are normal.
- 3) Check if transformer is bad.
- 4) Check if LCD's bright adjust and connection are normal.
- 5) Check if computer main board is normal.

6.2.3 System's control disorganize

- 1) Not correct operation.
- 2) The switch power's anti-jamming ability descend.
- 3) System's work circumstance become bad.

6.2.4 User's program lose

The DC battery on system main board can insure user's program and parameter don't lose. When system isn't used for half year or system has been used for over two years, the battery maybe invalidate, therefore, should exchange battery.

6.2.5 Machining precision is bad

- 1) CNC machine's reverse interval would change after using for a period of time, it needs to revise on time.
- 2) Best to revise base point before machining in order to insure the start point's precision.
- 3) Machining speed and cutting depth is improper.
- 4) Machine connector's prick melt falls off.
- 5) Tool isn't tightened.
- 6) Piece clamp isn't good.
- 7) Tool's giving up isn't equality because piece's dimension isn't uniformity.
- 8) Machine problem.

Appendix: binary、decimal switch table (0 — 15)

decimal	binary D7D0		decimal	binary D7D0
0	0 0 0 0 0 0 0 0		8	0 0 0 0 1 0 0 0
1	0 0 0 0 0 0 0 1		9	0 0 0 0 1 0 0 1
2	0 0 0 0 0 0 1 0		10	0 0 0 0 1 0 1 0
3	0 0 0 0 0 0 1 1		11	0 0 0 0 1 0 1 1
4	0 0 0 0 0 1 0 0		12	0 0 0 0 1 1 0 0
5	0 0 0 0 0 1 0 1		13	0 0 0 0 1 1 0 1
6	0 0 0 0 0 1 1 0		14	0 0 0 0 1 1 1 0
7	0 0 0 0 0 1 1 1		15	0 0 0 0 1 1 1 1

Note: Because of many kinds of reasons this Manual book may have some mistakes. Our company will provide the high quality service and the technical support for every customer.