K100Ti-B CNC System

USER'S MANUAL

Beijing KND CNC Technique Co., Ltd.

B05B-T00N-2101

K100Ti-B CNC System USER'S MANUAL B05B-T00N-2101

WWW.KND.COM.CN

CONTENT

I OVERVIEW

1 GENERAL

1.1 Introduction

K100Ti-B is a new generation CNC system designed by KND to satisfy the current situation of China for controlling economic lathes and two-axis machines. It adopts some new technologies such as 32-bit high-performance CPU, SLSI (super large scale integration), multiple layers PCBs, SMT etc. 7 inches TFT LCD is used to provide better performance. While maintaining lathes' common functions, K100Ti-B has been simplified to be smaller and more compact, as well as more reliable. K100Ti-B panel consists of a CNC operator's panel and a machine tool operator's panel, which simplifies machine linking. Operation interface is simple and concise, and the system has a high ratio of performance to price.

1.2 Specification Table

Function	Name		Specification
Axis	Controlled Axis Number		2 (X, Z)
	Simultaneously Controlled Axis Number		2
Command	Lease command unit		0.001 mm
	Least movement unit		0.001 mm
	Maximum command		±9999.999 mm
Feed	Rapid traverse rate		24000 mm/min (Max.)
	Feed rate	per min	1～20000 mm/min
		per revolution	0.0001～500.0000 mm/r
	Thread lead		0.0001～500.0000 mm
	Automatic acc./dec.		Supplied (Linear, exponential)
	Feedrate override		0～150%
Manual	Manual continuously feed Manually return to reference point Step feed		One axis at the same time ×1, ×10, ×100
	Hand wheel		Supplied
Interpolation	Positioning Linear interpolation Circular interpolation		G00 G01 G02/G03
Debug function	Dry running Single block funning		Supplied
Canned cycle	Internal/external cylindrical turning		G90
	Thread cutting cycle		G92
	Tapping cycle		G93

	Facing cycle	G94
Multiple canned cycle	Finishing cycle	G70
	Outer diameter rough turning	G71
	Rough facing cycle	G72
	Canned turning cycle	G73
	End peck drilling cycle	G74
	External/internal slotting cycle	G75
	Multiple thread cutting cycle	G76
	Enhanced thread cutting cycle	G78
Coordinate system and dwell	Dwell (second)	G04
	Coordinate system setting	G50
	Auto coordinate system setting	Supplied
Mode	MDI, Auto, JOG, Step, Edit	Supplied
Safety functions	Stored stroke check	Supplied
	Mask stored stroke check	Supplied
	Emergency stop	Supplied
Program storage and edit	Memory space and program number	32kB, 63
	Editing	Insert, modify, delete, copy
	Search function (Program no. sequence no. address, word)	Supplied
	Decimal programming Optional block skip	Supplied
	Electronic disks	Supplied, 6 regions
Display	TFT LCD	800×480, 7 inches
	Position information, program, tool offset, alarm, diagnosis data, debug, parameter	Supplied
	Graph, workpiece count	Supplied
M, S, T	I/O points: 20/16	Input: 4(panel)+16(machine)
	Miscellaneous function	M + 2-digit
	Spindle function	S + 2-digit
	Analog spindle (s+4digit) Constant surface speed cutting	Supplied (8-digit D/A)
	Tool function	T01~T08
	Tool rest signal checking function	Supplied
Compensation function	Memory	±6, 16 group
	Input by counting	Supplied
	Input by measuring	Supplied
	Type C tool compensation function	Supplied
	Backlash compensation	Supplied

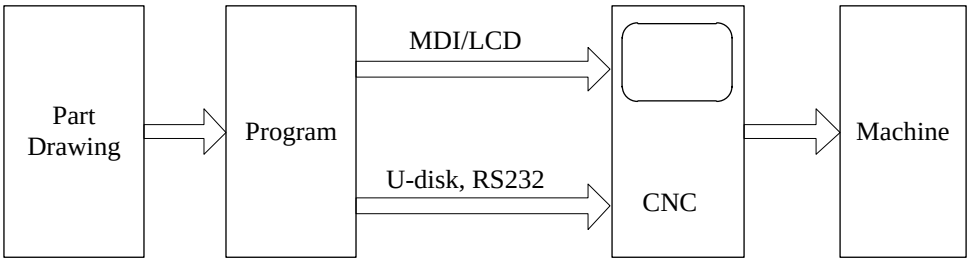
Switch	Program switch Parameter switch	Supplied
Communication	RS232	Supplied
	U-disk	Supplied
Other function	Circular radius R setting	Supplied
	Electronic gear ratio	Supplied
	Remember workpiece coordinate	Supplied
	Back too rest function	Supplied
	Start execution at random position	Supplied
	Rotary axis function	Supplied
	Clock and machining time	Supplied
Optional function	Additional machine operator's panel	Optional

1.3 General Operations

General procedures about using CNC machines are as follows:

1 Make machining programs according to your part drawing. Section II-PROGRAMMING in this manual describes how to make programs using K100Ti-B.

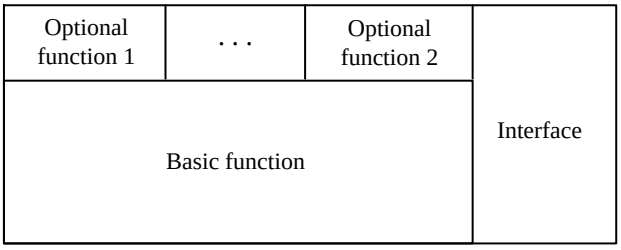
2 Transmit your program into CNC. Install proper tools and workpiece on machines. Execute your program in CNC, tool will move following programmed paths. In this manual section III-OPERATION describes the details about how to operate a CNC machine.



3 Section V-CONNECTION describes the dimension and connections information about using K100Ti-B. It also provides some information about maintenance.

1.4 About This Manual

1 Functions of CNC machine are not only determined by CNC system, they are also associated with machine structure, electric cabinet, servo system and some other elements.



As described in above figure, CNC system is composed of basic functions, optional functions and interfaces. For different machines, the functions used actually may vary. Before operating the machine equipped with K100Ti-B, read the operation manual provided by machine tool builder firstly.

2 This manual describes all the basic functions and optional functions of K100Ti-B. To achieve the best machining performance, operators should read operation manual form machine tool builder besides of this manual.

3 Using internal standard PLC which is able to satisfy multiple applications. If some special applications are needed, please contact machine tool builder.

4 The explanations in this manual (V1.0) are based on the following software and hardware versions:
Main board version: 0020I-0200-W01Z-0704
Software version: K100Ti-B_A02_120516
For the difference of some other versions, please refer to the supplementary specifications.

5 The price of K100Ti-B is different from different functions supported. Please contact KND for more information.

1.5 Safety Cautions

Transport and Storage

- Product packaging boxes can’t stack more than six layers.
- Don't climb, stand on and lay heavy load on the packaging box.
- Don't convey and drag the product by means of the cable that is connected with the product.
- Never collide or scratch the panel or screen.
- No humidity, insolation and drench on the packaging box.

Unpacked Check

- Unpack the goods to check that the product is identical with the ordered one.
- Check parts to be damage free due to the transit.
- Check parts to be complete and free from damage against the packing list.
- For any product model unconformity, missed parts and damage, you should contact KND promptly.

Connection

- The connection and check shall be done by qualified professional engineer(s).
- The product shall be grounded reliably and the grounding resistance shall be less than 4Ω, do not replace the grounding line by the zero (neutral) line.
- Connection must be correct and firm to avoid any malfunction and accident.
- You must connect a surge absorbing diode with the product according to the specified direction; otherwise the product may be damaged.
- You must power OFF before opening the back cover board, inserting and pulling off the plug.

Maintenance and Repair

- You must power OFF the system before repairing or replacing any component.
- When any short circuit and overload occurs, you must check the reason for that, and the system can be restarted only after the trouble is remedied.

- You should wait one minute to power ON after power OFF, frequent power ON and power OFF is forbidden.

★CAUTIONS

Electronic disks are available. It is recommended to store the proper parameters in electronic disks after debugging. If parameters or data are changed which resulting in the machine working unexpectedly, perform the restore operation to make system recovering quickly. Please refer to section III-OPERATION for more information.

Structure and specifications of K100Ti-B may be changed according to product upgrading.

II PROGRAMMING

1 COORDINATE SYSTEM

K100Ti-B CNC system is able to control two axes (X-axis and Z-axis) to move simultaneously, so the tool can move along straight lines or circular. In absolute programming mode, the address word (X, Z) and the coordinates of the end point are programmed; in incremental programming mode, the address word (U, W) and the move distance between the previous position and the next position are programmed. The two types of address words can be specified in a same block as (X, W) or (U, Z).

NOTE

The dimension of X-axis is specified in diameter mode in this CNC system.

1.1 Absolute Programming

In absolute programming mode, the address words (X, Z) and the coordinates of the end point are programmed.

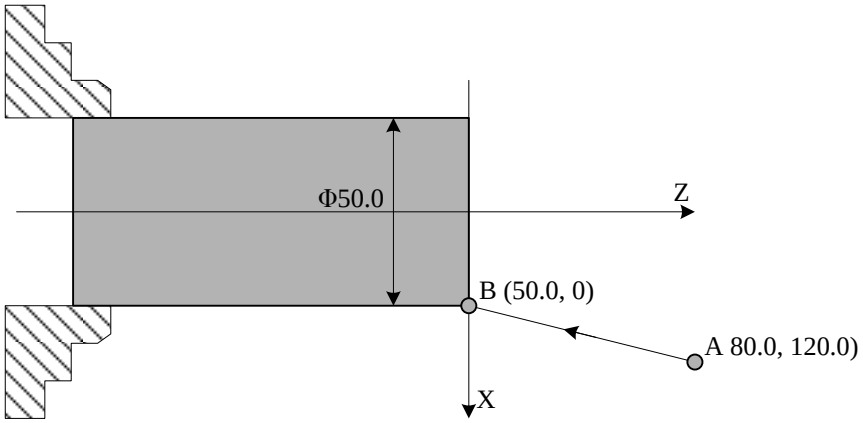


Figure II-1 Absolute commands and incremental commands

An absolute command for a tool to rapidly move from A to B (as shown in Figure II-1) is: G00 X50 Z0;

1.2 Incremental Programming

In incremental programming mode, the address word (U, W) and the move distance between the previous and the next tool position are programmed.

An incremental command for a tool to rapidly move from A to B (as shown in Figure II-1) is: G00 U-30 W-120;

1.3 Synthetical Programming

Absolute commands and incremental commands can be used in a mixed way in the system for convenience in programming and calculation. But in the same block, the same axis can only be expressed in one way. For example, you can program as (X, U) or (U, Z), but not as (X, U) or (Z, W).

In a mixed command that X axis employs absolute coordinate and Z axis uses relative coordinate for a

tool to move rapidly from A to B (as shown in Figure II-1) is: G00 X50 W-120;

1.4 Least Setting Unit

The minimum setting unit is 0.001 mm. The actual minimum movement of X axis is 0.0005 mm, and that of Z axis is 0.001 mm.

2 PROGRAM COMPONENTS

A set of CNC commands written to make a machine move as desired is called a program.

A program is comprised of multiple blocks; a block is composed of one or more words. One block is separated from another block via a block end code.

2.1 Program Format

A program consists of the following components: program start code, program number, program command section, program end command and program end code. In KND CNC system, both program start code and program end code is '%' as defined in applicable ISO standard. When a program is written, the program start code and program end code are added automatically. The program end code is always displayed, but program start code never.

Program format is usually as shown in Figure II-2.

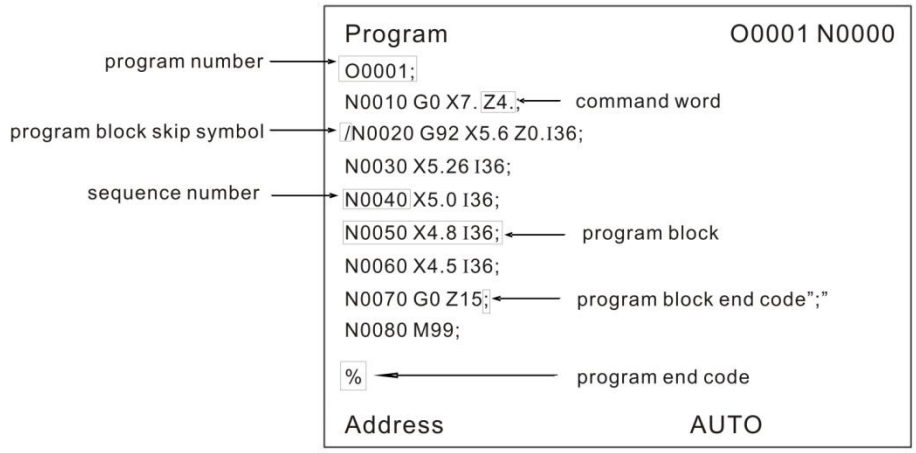


Figure II-2 Program components

2.1.1 Program number

The CNC memory can store many programs. Every program begins with a program number, ends with M30 or M99. A program number consists of address O followed by a four–digit number. A program number is assigned to each program in the memory to identify the program.

O ××××;

××××represents program number. (1~9999, the preceding zero will display without typing in)

2.1.2 Program command section

Every program consists of many commands. Its command unit is named block. Most blocks command a machine tool to act, such as a movement command or coolant on/off command. At the beginning of a block, it is a sequence number consisting of address N followed by a four–digit number (1~9999).

Address character	Function
O	Program number
N	Sequence number of blocks
G	Preparatory function
XZUW	Axis name
R	Specify the radius of an arc
IK	Specify the center of an arc
F	Feed rate
S	Spindle function
T	Tool function
M	Miscellaneous function
H	Specify the compensation number
P/X	Dwell time
P	Specify the number of sub-program
P	Number of subprogram repetitions
P/Q/R	Used by canned cycle commands

Sequence number

Sequence numbers can be specified in a random order, and any numbers can be skipped. It is permissible that no blocks or only important blocks be assigned with sequence numbers. In general, however, it is convenient to assign sequence numbers in ascending order according to the machining steps. In K100Ti-B CNC system, sequence numbers can increase automatically.

When parameter PA29 is not set to 0, S/N Automatically Increase function is valid. When pressing EOB key to end editing the current block, the S/N of the next block will generate automatically. The parameter value specified by PA29 is an increment. When a new S/N is inserted, the next S/N will increase from the inserted number by the increment.

Word

A block consists of words. A word consists of an address followed by some number. A sign may be prefixed to a number.

Word = Address + number (Example: X–1000)

For an address, one of the letters (A to Z) is used; an address defines the meaning of a number that follows the address. Table 2-1 indicates the usable addresses and their meanings.

The same address may have different meanings depending on the preparatory functions.

A block comprises several words; the word start is an English letter, which is called the address of the words. The functional category of the words is depending upon the address. In a program of this format, those words that are written in a previous block and will not change in current block will still be valid and need no writing again. Specifically, any modal G command e.g. G01 (refer to section 2.3) needs no rewriting if it has appeared in a previous block. In this format, the length of each word is not fixed. For example, in the word for dimension, it can go only with the valid numbers and with the preceding zero omitted (for example, G01 is equivalent to G1). Two blocks in a program are shown below:

- **Example**

N30 G01 X88.467 Z47.5 R50 S250 T0303 M03;

N40 X75.4;

The numbers of the words and characters in the above two blocks are highly different, but the functions of them are same except for the difference in X coordinates. The positions of the command words in the same block can be arranged in any way, for example, the "N30" in above example can also be written as:

- **Example**

N30 M08 T0303 S250 F50 Z47.5 X88.467 G01;

But in most cases, for the convenience of writing, inputting, checking and proofing, the words will be conventionally arranged in certain sequence in a block, that is, N→G→X→Z→S→T→M;

Program end

The following codes indicate the end of a program.

M30 indicates the end of the main program. Pressing  will run the main program from start.

M99 indicates the end of a subprogram. When M99 is performed, it will return to the main block that has called the subprogram.

2.2 Main Program and Subprogram

2.2.1 Main program

CNC Program is divided into two types, main program and subprogram. Normally, the CNC system runs according to the main program. However, as soon as M98 is encountered in the main program, the controller will run a subprogram. When M99 is encountered in a subprogram, the controller will return to run the main program.

The CNC memory can store 63 main programs and subprograms. We regard a program that never calls a subprogram also as a main program.

2.2.2 Subprogram

It can simplify the programming by making the same machining process of a workpiece as a subprogram stored into the memory. A main program can call a subprogram, which can also call another subprogram. When the main program calls a subprogram, it is regarded as a one–level subprogram call. Thus, subprogram calls can be nested up to two levels.

Format for subprogram calling

M98 P xxx yyyy

xxx: Times for calling a subprogram can be repeated (1~999), when no repetition times is specified, the subprogram is called just once.

yyyy: The called subprogram number must be 4-digit and the preceding zero can't be omitted. Range from 0001 to 9999.

- **Example**

M98 P51002;

This command specifies "Call the subprogram 1002 five times in succession." A Subprogram Call command (M98P_) and a move command can be specified in the same block. The example below shows

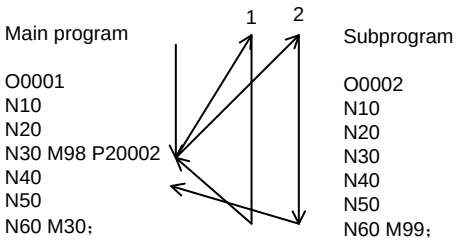
the subprogram calling after an X movement:

- **Example:**

X1000 M98 P1200;

Execution sequence

The execution sequence for a main program called a subprogram is shown in the following.



A subprogram calls another subprogram is identical with a main program calls a subprogram.

Special usage

If a subprogram runs alone, the machine tool can also work normally. When M99 is encountered in a subprogram, the controller returns to the start of the subprogram, and then cycle executes the subprogram until the 

key is pressed.

NOTE

1 If the subprogram number specified by address P can't be found, alarm (No. 78) occurs.

2 M98 command is invalid in MDI mode.

2.3 Optional Block Skip Function

The blocks with '/' symbol at its beginning are not executed when the BDT signal is set to 1, while they are executed when the BDT signal is set to 0.

Parameter Setting

043		SBDT						

Optional Block Skip is enabled when parameter SBDT is set to 1.

The input signal BDT controls the block skip switch to be enabled or disabled.

Input Interface

DGN 3					BDT			

The blocks with '/' symbol at its beginning are not executed when the BDT signal is set to 1.

3 PREPARATORY FUNCTION

The preparatory function consists an address G followed by two digits, it is to set the motion mode of the machine tool. G code command list is shown in Table 2-2.

Command format

G xx

xx: Command value(0~99)

G: Command address

G code is classified into two types: non-modal and modal. G code is also classified into seven groups: 00, 01, 02, 03, 04, 06, and 07.

- Non-modal G code: it is valid only in specified block code.
- Modal G code: it is valid until other G code in the same group occurs.

Example of modal G code

G01 and G00 locate in the same group:

G01 X__ F__; X axis feeds at the rate specified by F command, G01 is active

Z__; Z axis feeds, G01 is active

G00 Z__; G00 is active

G code	Group	Function
G00	01	Positioning (Traverse)
*G01		Linear interpolation (cutting feed)
G02		Circular interpolation CW
G03		Circular interpolation CCW
G04	00	Dwell, Exact stop
G28		Return to reference point
G32	01	Thread cutting
*G40	07	Tool nose radius compensation cancel
G41		Tool nose radius compensation (left)
G42		Tool nose radius compensation (right)
G50	00	Coordinate system setting
G70	00	Finishing cycle
G71		Multiple repetitive external diameter rough turning cycle
G72		Multiple repetitive rough facing cycle
G73		Multiple repetitive turning cycle
G74		Multiple repetitive end peck drilling cycle
G75		Multiple repetitive external diameter slotting cycle
G76		Multiple thread cutting cycle
G78		Enhanced thread cutting cycle

G90	01	O.D./I.D. cutting cycle
G92		Thread cutting cycle
G93		Canned tapping cycle
G94		Facing cycle
G96	02	Constant surface speed enabled
*G97		Constant surface speed disabled
*G98	03	Per minute feed
G99		Per revolution feed

NOTE

1 G codes marked with '*' are default codes at power on.

2 The G codes of group 00 are non-modal.

3 If a G code that is not listed in the Table 2-2 or a G code having no function is specified, alarm (No. 010) will occur.

4 Several different group G codes can be specified in a block. When several G codes from the same group are specified in a block, the G code specified last is valid.

5 G codes are expressed by group numbers.

3.1 Positioning (G00)

Tool rapidly traverses to specified coordinates in X and Z axis independently.

Command format

G00 X (U) _ Z (W) _;

Command description

X (U)_ and Z (W)_ are specified coordinates: -9999.999 ~ + 9999.999.

At G00 command, two axes move in speeds that are set respectively for them without influencing each other. As soon as any of the two axes reaches the commanded position, it stops; another axis continues to move until reaching the specified position.

At G00 command, the rapid traverse rate of each axis is specified by the parameter in the G00 command, and the feedrate that specified by address F is invalid. Using 

or 

key on the panel can modify G00 command's rapid traverse rate. The current overrides are displayed on the lower left corner of the Position page.

G00 is modal. G0 is equivalent to G00.

In order to avoid tool collision, when X axis and Z axis rapidly traverse simultaneously, take care that the position of the tool shall be within the safe range.

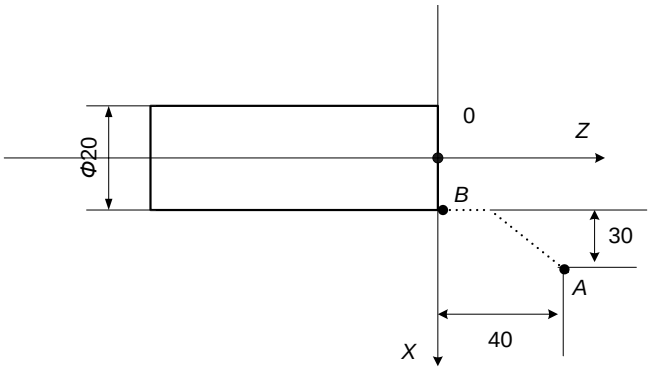


Figure II-3 Positioning

Example of G00 (as shown in Figure II-3): Tool moves from A to B at the rapid traverse rate.

Absolute programming: G00 X20 Z0;

Incremental programming: G00 U-60 W-40;

3.2 Linear Interpolation (G01)

The command enables a tool to move along a line from current point to a point specified by X(U) and Z (W) at the feedrate specified by F, two axes will simultaneously arrive at commanded position along a line.

Command format

G01 X(U)__ Z(W)__ F__ ;

Command description

1 X(U) _ Z(W) _ are specified coordinates: -9999.999 to + 9999.999.

2 F is modal.

3 The G01 command can set the linear interpolation along one of the two axes.

4 The feedrate override can be adjusted by pressing 

or 

key on the panel. The adjustable range is (0%~150%).

5 G01 is equivalent to G1.

Example of G01 (as shown in Figure II-4): Tool moves from A to B along a line at the feedrate specified by F.

Absolute programming: G01 X40 Z-30 F100;

Incremental programming: G01 U20 W-30 F100;

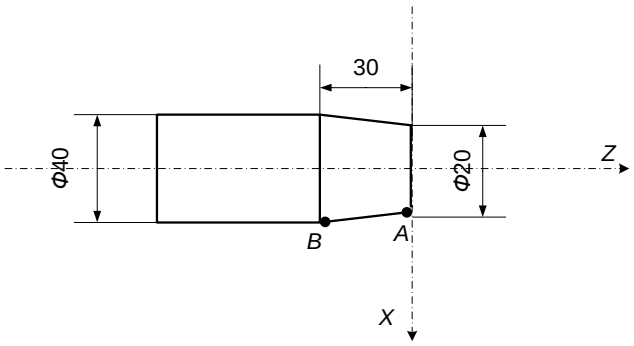


Figure II-4 Linear interpolation

3.3 Circular interpolation (G02, G03)

The command enables a tool to cut along a circular.

Command 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_ ;

Command description

G02,G03 command words are described in Table II-1.

Table II-1 G02/G03 command words

Description		Command	Meaning
Rotation direction		G02	Clockwise direction (CW)
		G03	Counterclockwise direction (CCW)
End point position	Absolute Value	X, Z	End point position in the work coordinates system
	Incremental Value	U, W	Distance from start point to end point
Distance from start point to center		I, K	I: X axis distance from the start point to the center of a circle. K: Z axis distance from the start point to the center of a circle.
Radius of circular		R	Radius of circular.
Feedrate		F	Feedrate along the circular tangent direction

1 I, K, R range: -9999.99 to +9999.99

2 The definition of CW and CCW are different in different tool rest systems as shown in Figure II-5.

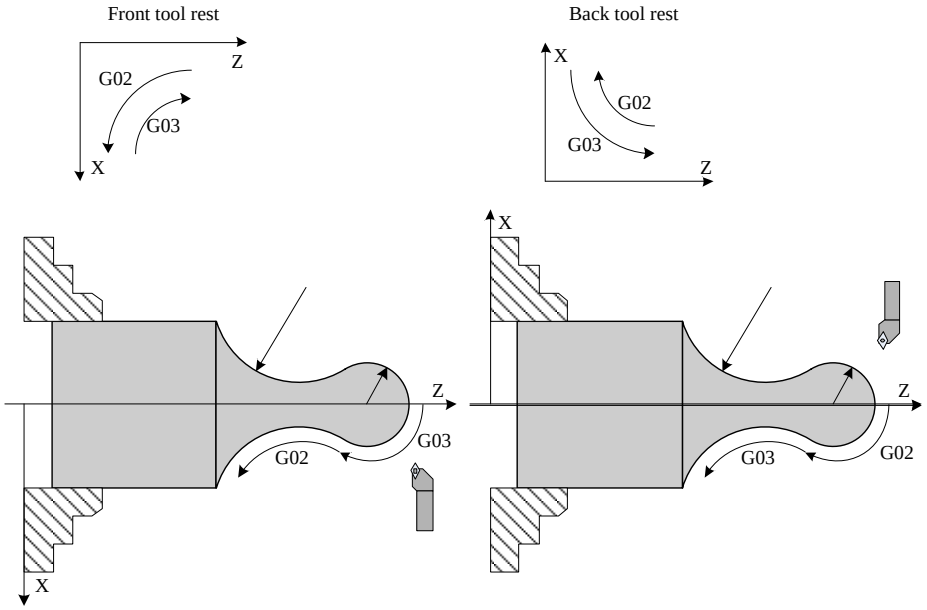


Figure II-5 Circular interpolation

3 The end point position is specified by address X, Z (absolute value) or U, W (increment value). Arc center can also be specified by I and K; I means X axis distance from the start point to the center of an arc, K means Z axis distance from the start point to the center of an circular as shown in Figure II-6. I and K must be signed according to the direction.

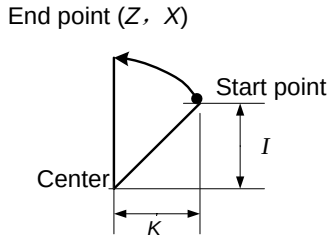


Figure II-6 Center position of circular

4 Besides I and K, circular center can also be specified by radius R. When the circular angle is more than 180°, the circular center cannot be specified by radius R. If I, K and R occur simultaneously in one block, R is valid rather than I and K.

5 Circular interpolation feedrate is specified by F, and is the tool speed along the circular tangent direction.

6 I0 and K0 may be omitted.

7 I and K are used to set circular radius.

8 When the circular center is specified by I or K, even if there is an error at start point and end point, no alarm will occur; when the circular center is specified by R, when there is an error at start point and end point, alarm will sure occur, so usually I and K are used to set circular radius.

Example of G02 (as shown in Figure II-7): Tool moves from A to B along circular. R=25.

	Absolute command	**Increment command**
I, K specified circular center	G02 X50 Z-20 I10 K-5;	G02 U20 W-20 I10 K-5;
R specified circular center	G02 X50 Z-20 R25;	G02 U20 W-20 R25;

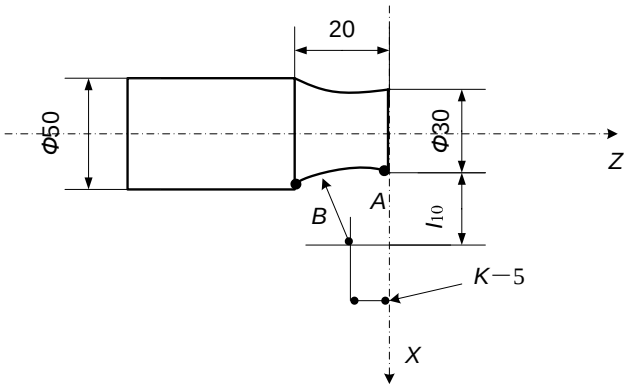


Figure II-7 Circular interpolation

3.4 Dwell (G04)

By setting a dwell, the execution of the next block is delayed with the specified time.

Command format

G04 P_; or G04 X_; or G04 U_ ; or G04

- P_: take ms as unit to command dwell time, in range 0.001~99999999 ms.
- X_U_: take sec as unit to command dwell time, in range 1~99999.999 sec.

Example

Tool G04 X 1; Dwell 1 second

G04 P1000; Dwell 1 second

G04 U 1; Dwell 1 second

Special Usage

G04 command without set value can be regarded as an exact stop command. A G04 command without set value can eliminate sharp-angled overcutting. An example of G04 command eliminating sharp-angled overcutting is shown in Figure II-8.

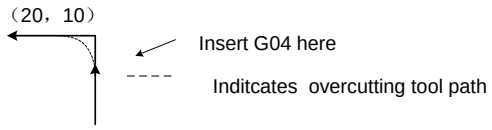


Figure II-8 G04 special usage

Example of G04 command eliminating sharp-angled overcutting

……

N150 G01 X20 Z10 F100;

N160 G04 ; Avoid overcutting

N170 G01 W-10;

……

NOTE

Setting SMZ of parameter PA03 to 1 can eliminate sharp-angled overcutting as well.

3.5 Automatic Return to Reference Point (G28)

A machine tool reference point (machine zero point) is a fixed position. When a machine zero point exists, it is the machine tool's reference point; When a machine zero point does not exist, a float zero point set can be regarded as the machine tool's reference point. Returning to reference point is either via manual Home mode or via G28 command (two axes automatically).

The machine moves from current position to middle point specified by X(U)_ Z(W)_ at rapid traverse rate, then returns to reference point.

Command format

G28 X(U)_ Z(W)_ ;

- X(U)_ Z(W)_ is the set middle point for traversing from current point to reference point .

Process for returning to reference point is as shown in Figure II-9.

Move tool rapidly to the middle point of the commanded axis (A→B).

Rapid positioning from the middle point to the reference point (B→R).

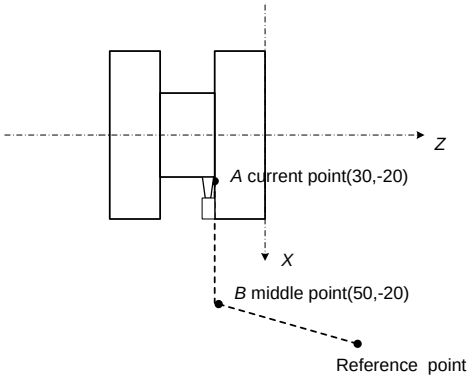


Figure II-9 Motion for automatically returning to reference point

If no manual return to reference point is executed after power ON, G28 command for moving from middle point to reference point is identical with motion of manual return to reference point.

Only the middle point of one axis is specified in G28 command, and the axis returns to reference point, and the other axis does not return to reference point.

Example

G28 X50 Z-20; or G28 U20 W0; (as shown in Figure II-9)

3.6 Work Coordinate System Setting (G50)

3.6.1 Coordinate system setting

Command format

G50 X __ Z __;

Command function

To set the absolute coordinates of current position so as to build work coordinates system. After executing this command, the system will take the current position as zero point of the program, and the machine

will return to this position at homing. Once the coordinates system is set, the positions of absolute command in the following commands are expressed by the coordinates in the coordinates system, until G50 sets a new work coordinates system.

Command description

1 G50 is non-modal.

2 In the compensation state, if G50 is used to set coordinates system, the position before compensation is G50 set work coordinates system position. Usually tool compensation is cancelled before running the program. KND system automatically cancels tool compensation after returning to reference point

3.6.2 Coordinate system translation

Command format

G50 U __ W __;

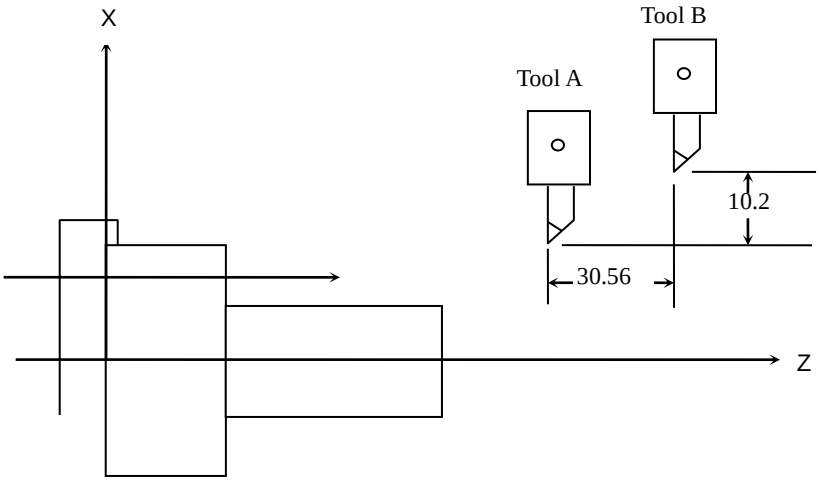
Command function

After executing the G50 command such as, the tool coordinate values is changed to the values in this command, which is based on the newly established coordinate system.

Whether the value of X/u is diameter or radius depends on the programming mode of diameter-specified or radius-specified.

(Example) Coordinate system translation from Tool A to Tool B.

G50 U20.4 W30.56 ； (diameter-specified)

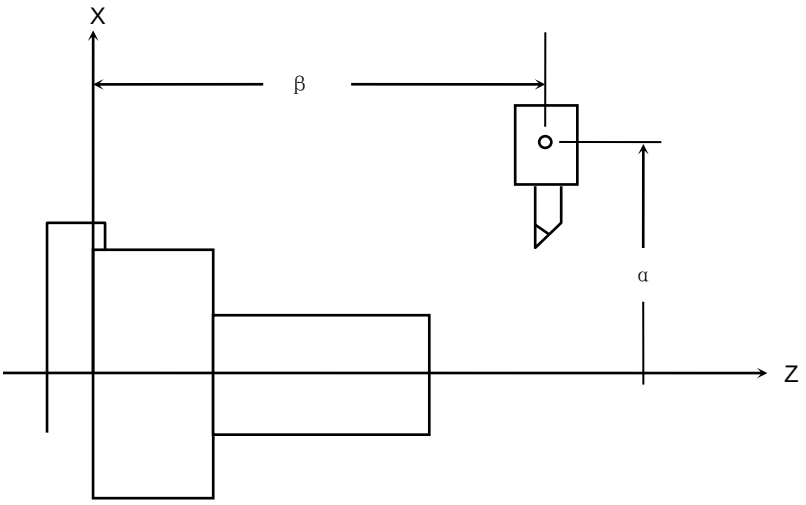


3.6.3 Automatically set the work coordinate system

Given P013 is set to α and P014 is set to β, the coordinate system is established automatically after the machine tool returns to the zero point manually. The coordinate of the reference point on the turret or the tool nose is: X=α, Z=β。

It is equivalent with the following command in program:

G50 Xα Zβ ；



3.7 Thread Cutting (G32)

G32 enables to cut constant pitch straight threads, taper threads and end threads.

Command format

G32 X(U) __ Z(W) __ F(I)__;

Command description

1 G32 is a modal command.

2 X(U) and Z(W) is thread end point coordinates; for straight threads cutting, X(U) is omitted; for end threads cutting, Z(W) is omitted; for taper thread cutting, no X(U) or Z(W) is omitted.

3 F__ is metric pitch, that is tool displacement of spindle per revolution in the axial direction of workpiece, unit: mm, range: 0.0001~500.0000.

4 I__ is the number of teeth of an inch thread, which is spindle revolution counts of one inch tool displacement in the axial direction of workpiece, unit: tooth number per inch. Range:0.060~254000.000. I is non-modal.

5 In general, thread cutting needs machining many times along the same tool path from rough to finish cutting. As the thread cutting starts when the signal of one revolution from the position encoder on the spindle is detected, even if cutting multiple threads, the cutting point on the job circumference is same, thread cutting path of the job is same as well, but this requires constant spindle speed from rough to finish turning. Error will occur when spindle speed varies. At the start position and end position for thread cutting, as a result of acceleration/deceleration, lead of thread is incorrect. Take the factor into account, the commanded thread length should be longer than required thread length.

6 Feedrate override is invalid during thread cutting and is fixed at 100%.

7 Don’t stop spindle during thread cutting, which is quite dangerous because spindle-pause will result in the cut depth increase sharply. Dwell function is invalid at thread cutting.

8 When thread cutting is performed in Single Block mode, the tool stops after execution of the first non-thread cutting block.

9 If both the previous and current block are for thread cutting, cutting will start immediately without

detecting the one–revolution signal.

- G32 W-20 F3 (one–revolution signal is detected before this block.)
- G32 W-30 F2 (one–revolution signal is not detected before this block.)

10 For the applications when cutting a big pitch thread, set LW to 1.

Example

- **Straight thread cutting (as shown in Figure II-10)**

Thread lead: 4mm; &1＝3mm (thread acceleration phase,&1≥3mm); &2＝1.5mm (thread deceleration phase,&2≥1.5mm);

G00 U-62.0;

G32 W-74.5 F4.0;

G00 U62;

W74.5;

U-64; (cut in 1mm again for the second cut)

G32 W-74.5;

G00 U64.0;

W74.5;

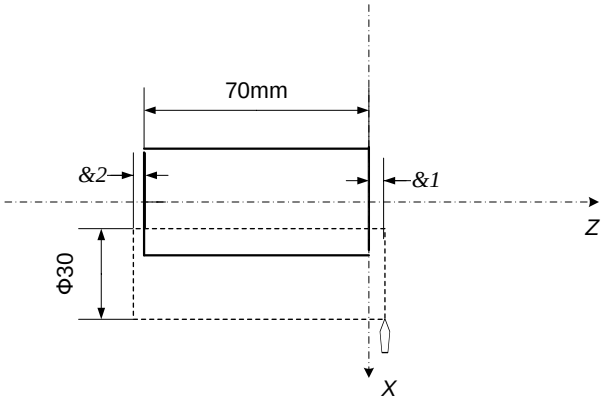


Figure II-10 Straight thread cutting

- **Taper thread cutting (as shown in Figure II-11)**

Thread lead : 3.5mm ;&1＝3mm ;&2＝1.5mm;

Depth of cut in X direction is 1mm (cut twice)

(Metric input, diameter programming)

The program is as shown in the following:

……

G00 X12 Z3.0;

G32 X41.0 Z-41.5 F3.5;

G00 X50;

Z3;

X10;

G32 X39 Z-41.5;

G00 X50;

Z3;

……

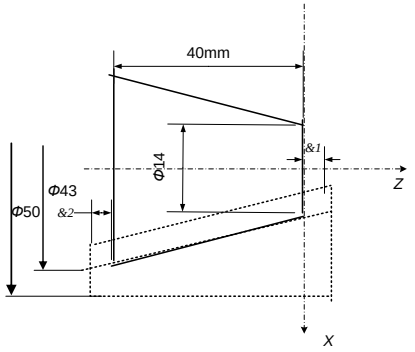


Figure II-11 Taper thread cutting

3.8 Thread Cutting Cycle (G92)

G92 command makes it possible to execute straight thread cutting (as shown in Figure II-12): cutting(①) — cutting thread (②)— retracting tool (③)—returning to start point for machining thread(④).

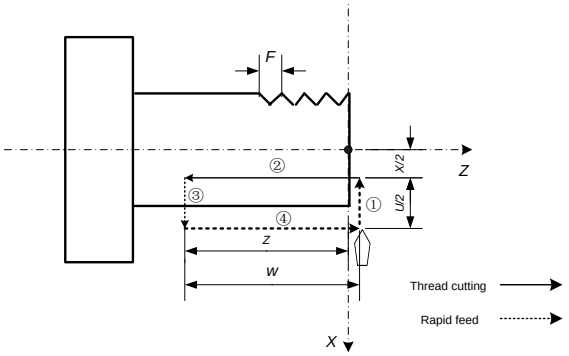


Figure II-12 Straight thread cutting (G92)

Command format

G92 X(U)__ Z(W)__ F(I)__ R__L__J__K__P__;

Command description

1 G92 is a modal command.

2 X(U)__ Z(W)__ is coordinate of the thread end point.

3 F is metric screw-pitch, which is tool displacement of spindle per revolution in the axial direction of workpiece, unit: mm; range: 0.0001~500.0000.

4 I is the number of teeth of an inch thread, which is spindle revolution counts of one inch tool displacement in the axial direction of workpiece, unit: tooth number per inch. Range:0.060~254000.000. I is non-modal.

5 R is radius difference in X direction between thread cutting start point and thread cutting end point, which as shown in Figure II-13. In the X direction, R is negative when cutting start point value is less than cutting end point value, otherwise R is positive. Using R can machine taper thread.

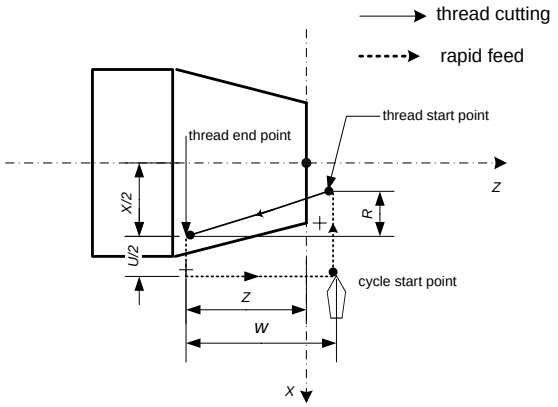


Figure II-13 Cutting cycle for taper external thread

6 L specifies no. of starts of a thread, L is a modal value, range: 1 to 100.

7 J/K: specifies proportion of X to Z axis when thread cutting run outs, J/K is a modal value in range 1~4. When setting other values, thread cutting retreat angle is fixed at 45 °.

8 P specifies thread cutting retreat width, unit: 0.1 pitch, modal, setting rang: 1~225. Executing P__ of G92 can change the value of P'28, which keeps invariable after power OFF.

9 Acc./dec. control in thread cutting cycle

10 As the result of exponential acc./dec. control, pitch will be uneven at the end of the thread. The higher the spindle speed, the longer the uneven thread pitch. Reducing exponential acc./dec. time constant can reduce error, however, which will cause step motor to be locked. In order to resolve this problem, we can do as follows:

- Select linear acc./dec. control in X/Z axis;
- Select retreat at G00 rapid traverse rate in X axis.

11 When cutting big pitch threads or multiple threads, set LW to 1.

Parameter

041			ZG92L					

5 ZG92L Select the acceleration style of Z-axis when cutting thread using G92 and G76

0: Linear acceleration

1: Exponential acceleration

042							G92L	

1 G92L Select the acceleration style of X-axis when cutting thread using G92 and G76

0: Exponential acceleration

1: Linear acceleration

043				LW				

4 LW Select the optimized method when cutting big pitch threads

0: Invalid

1: Valid

055	G92XR	XG92P						

7 G92XR Select the chamfering method in X-axis direction when cutting thread(G92/G76)

0: Chamfering with optional angles (by specifying the J/K commands)

1: When chamfering, X-axis retracts at the rapid traverse rate

6 XG92P Select the retracting method in X-axis direction when cutting thread(G92/G76)

0: Tool retracts with the traditional method

1: Tool retracts in the X-axis direction at the rapid traverse rate

057	G92LINTX

Linear acc./dec. time constant when controlling X-axis in thread cutting

058	G92LINTZ

Linear acc./dec. time constant when controlling Z-axis in thread cutting

Precision control for thread cutting

At thread cutting, spindle speed stability influences greatly the thread precision. It is necessary to keep constant spindle speed to machine precision thread. It is possible to carry out precision thread machining through controlling concerned system parameter.

042						NTHD		LWN

2 NTHD Weather detecting the stability of the spindle rotation when cutting thread

0: Not detect

1: Detect. P59 and P60 are valid.

059	Revolution count of spindle

Circle number used when calculate the average speed of the spindle

060	Allowable fluctuation of the spindle speed when cutting thread

Example

- **Straight thread cutting by G92**

Shown in Figure II-14, F＝1.5mm

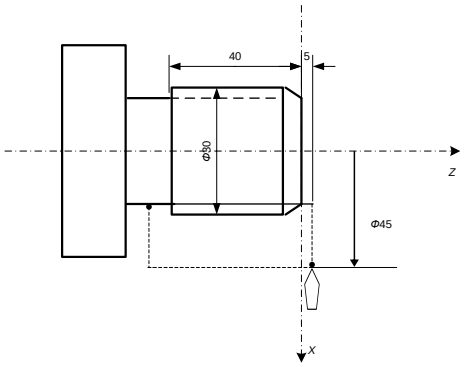


Figure II-14 Straight thread cutting (G92)

N10 M03 S××;

N20 T0101;

N30 G00 X45 Z5;

N40 G92 X29.2 Z-40 F1.5;

N50 X28.6;

N60 X28.2;

N70 X28.04;

N80 G00 X100 Z50;

N90 T0100 M05;

N100 M30;

- **Taper thread cutting by G92**

Shown in Figure II-15: pitch: 11 tooth/inch (taper＞1: 32)

N10 M03 S××;

N20 T0101;

N30 G00 X55 Z10;

N40 G01 X60 Z5 F100;

N50 G90 X66.25 Z-60 R1.875;

N60 G92 X66.88 Z-50 R1.4 I11;

N70 X66.9 I11;

N80 X67 I11;

N90 X67.4 I11;

N100 X67.6 I11;

N110 X67.8 I11;

N120 G00 X100 Z50;

N130 T0100 M05;

N140 M30;

NOTE

The lead specified by I is non-modal, it must be input in every thread cycle block.

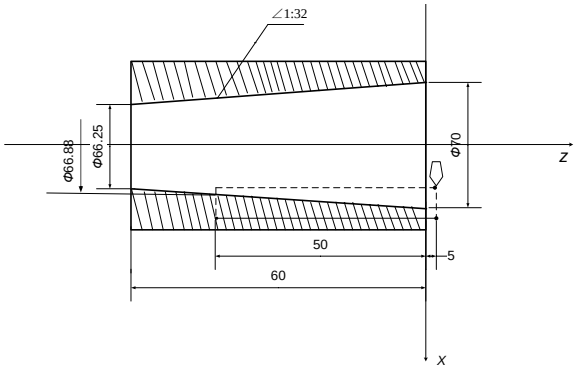


Figure II-15 Internal taper thread cutting cycle (G92)

- **multi-start thread cutting by G92**

O1234;

……

G92 X50 Z-100 F6 L5; at X50, machining five threads

X48.5; at X48.5, machining five threads

X45; at X45, machining five threads

G00 X100 Z100;

……

3.9 Multiple Thread Cutting Cycle (G76)

G76 are used to cut straight thread and taper thread with retreat function. Through several rough thread turnings and finish thread turnings, thread cutting of specified tooth height (total depth of cut) can be carried out. Thread cutting with cutting edge at one side can be realized. The tool engagement decreases

gradually, which is good for protecting the tool. The machining paths are as shown in Figure II-16.

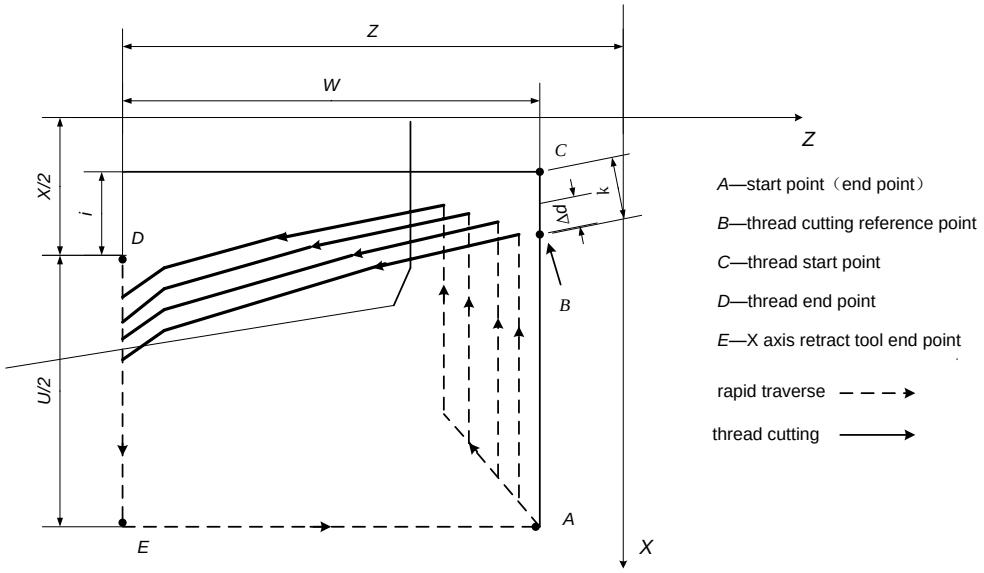


Figure II-16 Canned thread cutting cycle (G76)

Cutting method in detail is as shown in Figure II-17

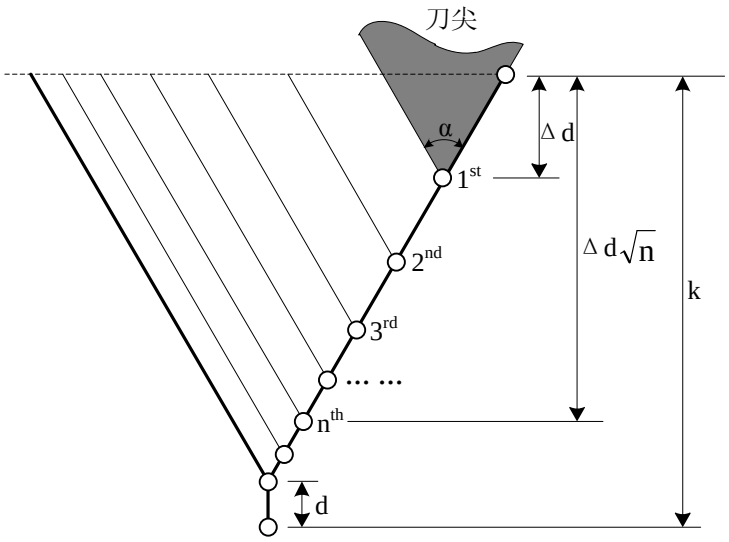


Figure II-17 Feed method when cutting thread (G76)

Command format

G76 P(m)(r)(a)Q(Δdmin)R(d);

G76 X(U) Z(W) R(i) P(k)Q(Δd) F(I) ;

Command description

1 m——Repetitive counts of finish machining (1 ~ 99). This value is modal. Also this value can be specified by the parameter P'27, and the parameter is changed by program command.

2 r——Chamfering amount for the thread. When the thread lead is expressed by L, the value of r can be set from 0.01L to 9.9L in 0.1L increment (2–digit number from 00 to 99). This value is modal. Also this value can be specified by the parameter P'28, and the parameter is changed by program command.

Chamfering length is 1 pitch when r is equal to 10.

3 a——Angle of tool nose (tip). Six angles: 80°, 60°, 55°, 30°, 29°and 0°can be selected. It is specified by a 2–digit number. This value is modal. Also this value can be specified by the parameter P'29, the parameter is changed by program command.

- m, r and a are specified by address P at the same time.
- When m=2, r=1.2L, a=60°, the setting is as follows (L is lead of thread).
- Example: P 02 12 60

4 Δdmin: Min. depth of cut (specified by the radius value). When $\Delta d \times \sqrt{n} - \Delta d \times \sqrt{n-1}$ (the depth of cut of one cycle) is smaller than Δdmin, the depth of cut is clamped at Δdmin. This value is modal. Also this value can be specified by the parameter P'30, and the parameter is changed by program command. Unit is μm.

5 d——Finishing allowance. This value is modal. Also this value can be specified by parameter P'31, and parameter is changed by program command. Unit is mm.

6 X(U) Z(W) ——the end position for thread cutting.

7 i ——radius difference of threaded sections. If i = 0, it is to cut straight thread. Unit is mm.

8 k ——Height of thread (Value in X direction is specified by the radius). Unit is μm.

9 Δd ——Depth of cut in 1st cut. Unit is μm.

10 F(I)——Pitch of thread (same as G32).

11 By using G76 machining cycle, cutting is performed with cutting edge at one side, and the load on the tool nose (tip) is reduced.

12 Making Δd for the first cut, and $\Delta d \times \sqrt{N}$ for the nth cut, cut amount each time is held constant. Four patterns are considered corresponding to the sign of each address. Also the internal thread cutting is possible. For the thread cutting in Figure II-16, only the feed rate between C and D is specified by F, and the feed rate for other path is rapid traverse rate. The signs of incremental dimensions for the Figure II-16 are as follows :

- U, W— minus: determined by the direction of the tool path A to C and C to D.
- R(i)—minus: determined by the direction of the tool path A to C.
- P(k)—plus: always positive
- Q(Δd) — plus: always positive

13 Cautious about cutting thread is same as those on G32 cutting thread and G92 cutting thread cycle.

Example

Canned thread cutting cycle (as shown in Figure II-18)

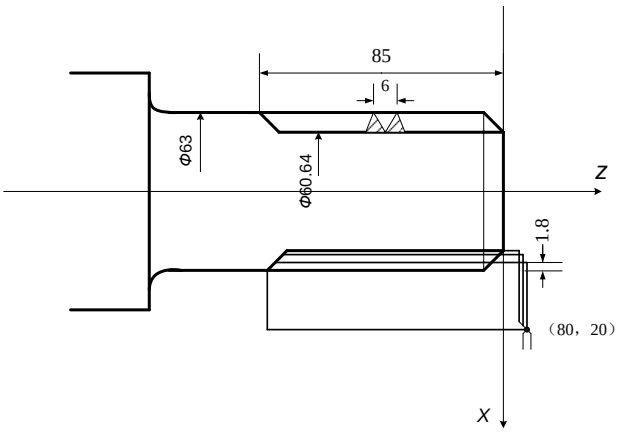


Figure II-18 Canned thread cutting cycle (G76)

```
O0022;
N10 M03 S××;
N20 T0101;
N30 G00 X80 Z20;
N40 G76 P011060 Q100 R0.1 ;
N50 G76 X60.64 Z85 P3680 Q1800 F6.0 ;
N60 G00 X100 Z50;
N70 T0100;
N80 M05;
N90 M30;
```

3.10 Enhanced Thread Cutting Cycle (G78)

G78 is divided into two commands. The first is a command for setting parameters; the second for machining.

3.10.1 First command for setting parameters

Command format

G78 P (a b cc) R(r);

a: Depth of cut per pass. Range 0~2.

b: Cutting mode. Range 0~3.

cc: Tip angle of the thread tool, Range 0~90

Command description

1 Sharp angle of thread. Range 0～90.

2 Depth of per cutting operation:

- 0——Feeding at equal distance;
- 1——Feeding decreasingly for each cut

$$\Delta d = (\sqrt{n} - \sqrt{n-1}) \times R / \sqrt{L}$$

(Δd: feed at nth cuts; n: nth feeding, n≤L; L: Cycle time; R: Total depth of cut)

- 2——If the stock removal is too high at first pass in decreasingly feeding, divide the first pass into two passes.

3 Cutting mode:

- 0——Tool edge cuts along middle line of the thread profile.
- 1——Tool edge cuts along the left of the thread profile.
- 2——Tool edge cuts along the right of the thread profile.
- 3——The tool edge cuts alternately along the left and the right of the thread profile. The first pass cuts along the middle line, and when the depth of cut is specified as 2, both passes will cut from the middle line. The last pass for finishing also cuts from the middle line of the last rough cut to correct the profile from two sides.

4 r : Allowance for finishing, it is modal, can be set by parameter P31. The finishing allowance is subtracted from the last pass of rough cutting. Unit: mm.

3.10.2 Second command for machining

Command format

G78 X(U) _ Z(W) _ F/I_ E_ J_ K_ R_ H_ L_;

Parameter description

1 X(U)——Coordinate of external diameter at the end of thread in X axis, programmed in absolute or incremental way. The default is straight thread.

2 Z (W)——Thread length, programmed in absolute or incremental way.

3 F/I ——Pitch. F- metric thread (mm/lead or inch/lead); I – inch thread (tooth/inch)

4 E——Retreat distance in X direction. Programmed in diameter. The sign means the direction for the retreat. Unit: mm.

5 J ——Retreat correction in Z axis. That is, the retreat amount in advance, positive.

6 K——Screw-in distance in X direction. Programmed in diameter. The sign means the direction for screwing in.

7 R ——Thread tooth height (diameter difference between the thread root and thread crest).

8 H ——number of thread starts. Range: 1≤H≤100; if out of the range, H=1.

9 L——Times of cycle. Default L=1.

Command description:

1 As for the command programming range and units for F/I, refer to G32 command.

2 2th page of “Parameters” will be visible only by pressing 

key.

- P’32— Depth for each cut
- P’33— Selection of cutting mode
- P’34— Speed for screw in and retreat. Range is 200~5000 mm/min. When the setting value is less than 200 mm/min, it will automatically adjust to 2500 mm/min.

3 When the J value is not programmed, Z-axis retreat amount will be specified according to the default value. By programming J value, equal-pitch retreat can be achieved. J means the Z-axis retreat length, that is, when the distance from the tool nose to the thread end is J, X axis starts to retreat.

4 When K screw-in value is programmed, the tool nose must be located beyond the workpiece surface within a distance ≥K value, otherwise it will result in overcut and cause tool nose damage.

5 When both K and E are programmed, they shall be of reverse symbols, otherwise ALARM (№62) will occur.

6 When K is programmed without E, it is deemed that E = －K.

7 Acceleration/deceleration time constant of X axis and Z axis is same as the time constant of G92. Both time constants are specified by parameter P57 and P58.

8 If only G78 X_ Z_ F/I is programmed, thread cutting with single tool will be carried out, and no tool retraction will be conducted after cutting.

9 When K or E is not programmed, one among L, H and R should not be programmed; otherwise ALARM (№62) will occur.

Example 1

Without screw-in and screw-out function (as shown in Figure II-19)

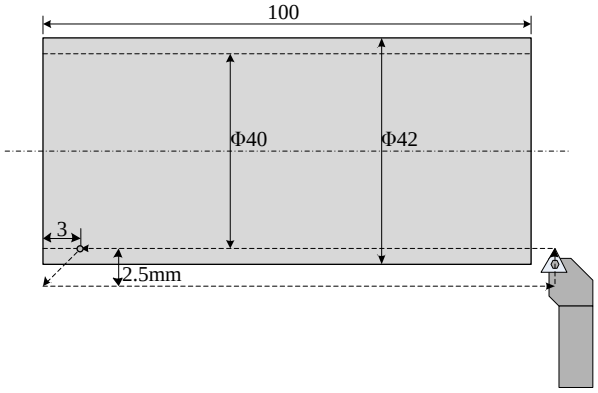


Figure II-19 Example 1 diagram

O0001；

M03 S500；

G98 G01 X40. Z0；

G78 P1060 R0.1；

(Feeding decreasingly and along the middle line, 60 °tool, finishing stock: 0.1mm)

G78 Z-100 F3.0 E5.0 J3.0 R2.0 H3 L5；

(Pitch: 3mm, Thread start number: 3, retract length: 3mm, tooth height: 1mm, cycle time: 5)

M30；

Example 2

With screw-in and screw-out function (as shown in Figure II-20)

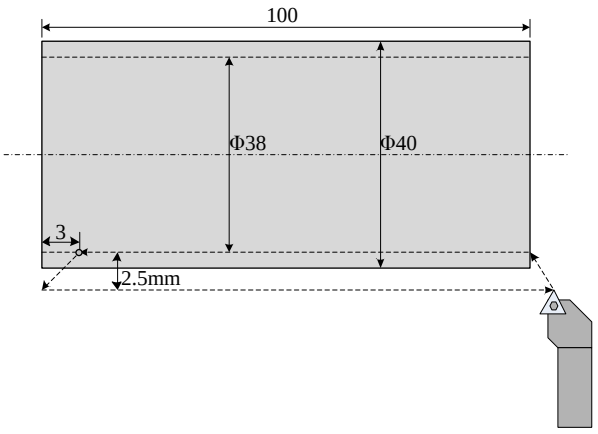


Figure II-20 Example 2 diagram

O0002；

M03 S500；

G98 G01 X45.0 Z0；

G78 P1060 R0.1；

(Feeding decreasingly and along the middle line, 60 °tool, finishing stock: 0.1mm)

G78 Z-100 F3.0 K-5.0 J3.0 R2.0 H3 L5；

(Pitch: 3mm, Thread start number: 3, retract length: 3mm, thread-in length: 5mm, tooth height: 2mm, cycle time: 5)

M30；

Example 3

Internal thread, without screw-in and screw-out function (as shown in Figure II-21)

O0003；

M03 S500；

G98 G01 X40. Z0；

G78 P1060 R0.1；

(Feeding decreasingly and along the middle line, 60 °tool, finishing stock: 0.1mm)

G78 Z-100 F3.0 E-5.0 J3.0 R2.0 H3 L5；

(Pitch: 3mm, Thread start number: 3, retract length: 3mm, tooth height: 2mm, cycle time: 5)

M30；

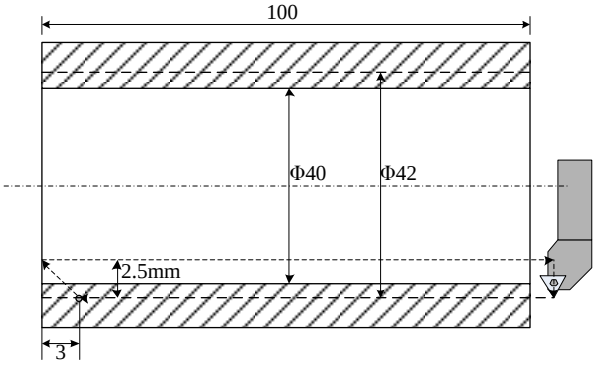


Figure II-21 Example 3 diagram

Example 4

Taper thread cutting, with screw-in function (as shown in Figure II-22)

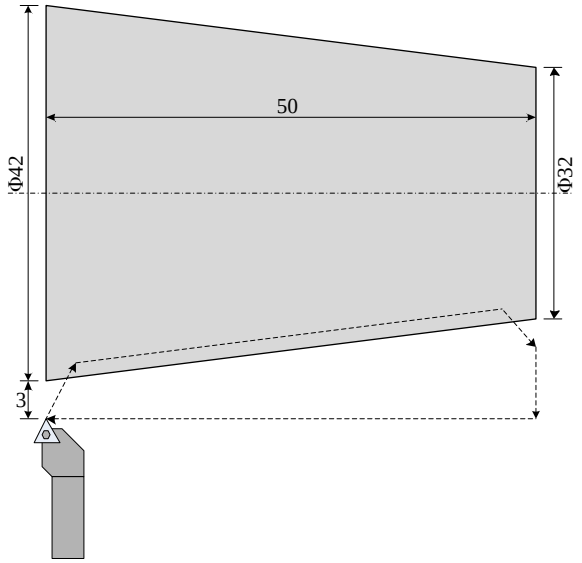


Figure II-22 Example 4 diagram

O0004；

M03 S500；

G98 G01 X48.0 Z0；

G78 P1060 R0.1；

(Feeding decreasingly and along the middle line, 60 °tool, finishing stock: 0.1mm)

G78 X32.0 Z50.0 F2.5 J3.0 K-6.0 R2.0 L5；

(Pitch: 2.5mm, single thread cutting, retract length: 3mm, thread-in length: 6mm, tooth height: 2mm, cycle time: 5)

M30；

3.11 Cylindrical Or Taper Cutting Cycle (G90)

From cutting start point, feeds in radial (X axis) direction, cuts in axial (Z axis or both X and Z axis) direction. The command makes it possible to finish cylindrical and taper cutting cycle.

Command format

G90 X(U)__ Z(W)__ F__; (cylindrical cycle) or

G90X(U)__ Z(W)__ R__ F__; (taper cycle)

Command description

1 G90 is modal.

2 X__ Z__ is the coordinate of end point for cylindrical face cutting. U__ and W__ are coordinate components of cylindrical face cutting end point relative to the cycle start point. Cylindrical cutting cycle is as shown in Figure II-23.

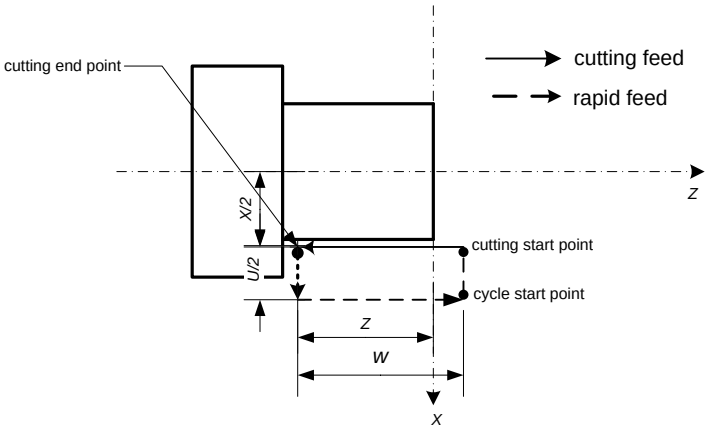


Figure II-23 Cylindrical cutting cycle

3 R is radius difference between cutting start point and cutting end point. Taper cutting cycle process is as shown in Figure II-24. Where the cutting start point coordinate is less than cutting end point coordinate, it is regarded as climb taper, R is negative; otherwise, R is positive.

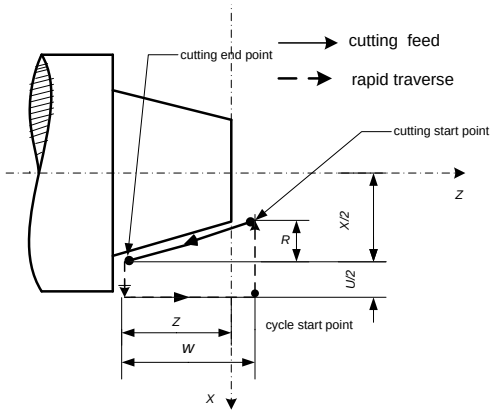


Figure II-24 Taper cutting cycle

Example

Cylindrical cutting cycle (as shown in Figure II-25)

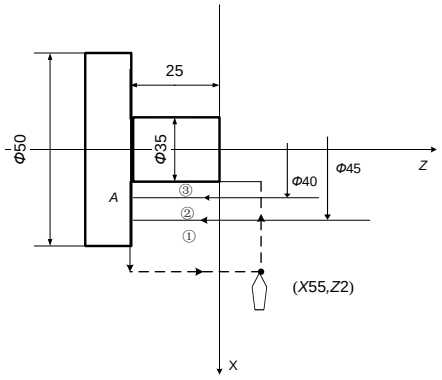


Figure II-25 G90 cylindrical cutting cycle

O0001;

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;

In order to improve the efficiency, the above program can be simplified as:

N50 G90 X45 Z-25 F100;

N60 G00 X47;

N70 G90 X40 Z-25;

N80 G00 X42;

N90 G90 X35 Z-25;

N100 G00 ;

Example

It shows the taper cutting cycle. (as shown in Figure II-26. Programmed by constant surface feed per revolution)

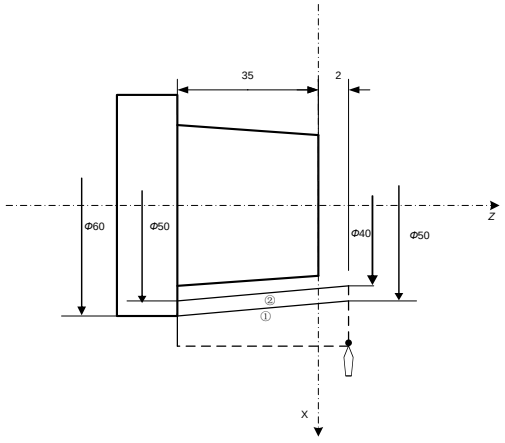


Figure II-26 Usage of G90

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; (R＝(D_{start} －D_{end}) / 2＝(40－50) / 2＝－5)

N80 X50;

N90 G00 G98 X100 Z100 M09;

N100 G97 S1000 T0100;

N110 M05;

N120 M30;

3.12 Facing Cycle (G94)

It is, from the cutting point, to execute axial (Z axis) feed and radial (Z axis or X, Z) cutting. The command makes it possible to finish facing or taper facing cycle.

Command format

G94 X(U)__ Z(W) __ F__; (facing)

G94 X(U)__ Z(W) __ R_ F__; (taper facing)

Command description

1 G94 is a modal command.

2 X, Z are facing end point coordinates, U, W are facing end point coordinate components relative to cycle start point. The cycle process is as shown in Figure II-27.

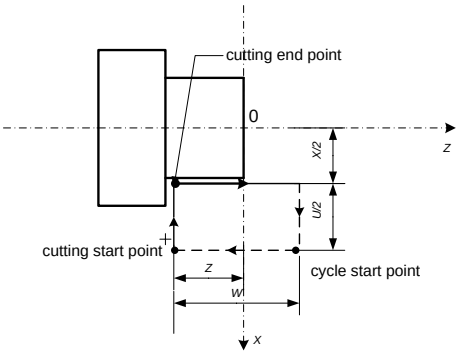


Figure II-27 Use of G94 facing cycle

3 R is displacement component in Z direction (facing start point to end point). The taper facing cycle process is as shown in Figure II-28. If the path is such that the Z axis is of negative direction, R is negative; otherwise R is positive.

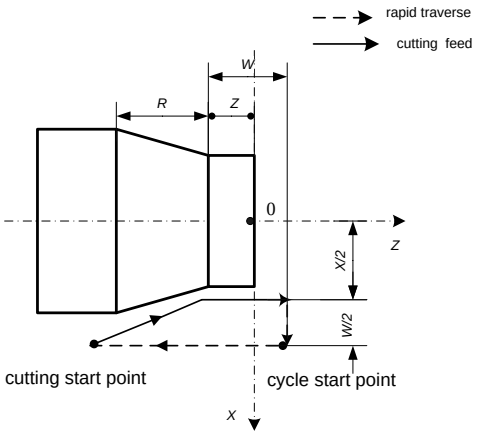


Figure II-28 Use of G94 (taper facing cycle)

Example

G94 facing cycle (as shown in Figure II-29)

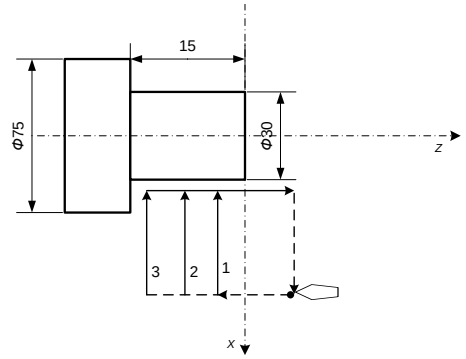


Figure II-29 Use of G94 (facing)

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;

For above program, every cycle returns to the start point, the outer diameter is cut repeatedly; in order to run efficiently, the program's cycling part can be changed as follows:

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 X Z;

Example

G94 taper facing cycle (as shown in Figure II-30)

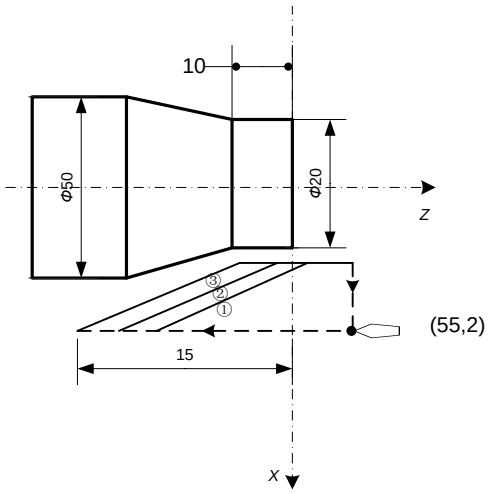


Figure II-30 Use of G94 (taper facing)

Segment of the program is:

N40 G01 X55 Z2 F200;

N50 G94 X20 Z0 R-5 F100;

N60 Z-5;

N70 Z-10;

N80 G00 X Z;

In N50 block: R＝－15－(－10) ＝－5mm

3.13 Canned Tapping Cycle (G93)

The command is for internal thread cutting cycle. The tool path is from the start point to the end point, then from end point to the start point. During the movement, Z axis moves one pitch as the spindle runs one revolution, so as to maintain same pitch as the tap, a thread slot is generated in the inner hole of the workpiece, the thread cutting for the inner hole can be realized in one pass.

Command format

G93 Z(W) __ F(I) __ ;

Command description

1 G93 is a modal command.

2 Z (W)—— coordinate of the Z axis end position.

3 F(I)—— pitch, same as G32.

4 Execution procedures: Z axis feeds in negative direction in a form for thread cutting. After moving to the coordinate specified by the program, the spindle stops automatically, after it stops completely, the spindle rotates automatically in specified reverse direction, and Z retracts to the start position. The spindle stops, and restores the rotation direction that is specified before the block.

5 If G93 is performed after Z axis runs in positive direction, the system will first carry out reverse backlash compensation because it is reverse movement. In this case, you should set parameter RVDL=0 of PA03. If the configured step motor blocks, you can set a lower reverse backlash compensation frequency, or command Z axis to move in negative direction before executing G93.

6 The setting of the spindle braking time will influence the reverse start time after stopping, please take care of this in the setting.

7 Z must be in negative movement, otherwise it generates ALARM (№12) 'G93 Format Error'.

8 Don't program X value, otherwise it generates ALARM (№12) 'G93 Format Error'.

9 The spindle must be started up before executing G93.

10 The spindle braking time of the machine is required to be short. At system preparation, it is assumed to be a movement of +50.000. When requiring the output spindle to stop, the movement length cannot exceed 50 mm.

11 The spindle speed is required to be not too high.

12 Acceleration or deceleration during tapping can be selected by the following parameter:

041				G93N				

4 G93N Select the acc./dec. style when performing the tapping operation (G93)

0: No acc./dec. control

1: Exponential acc./dec. control

NOTE

When acceleration/deceleration control is selected, if spindle speed changes, it will delay the change on thread. Therefore, when it requires high accuracy, you shall select no acceleration/deceleration. However, when a step motor is used, the spindle speed shall not be too high; otherwise, no acceleration/deceleration will block the motor.

3.14 Outer Diameter Rough Turning Cycle (G71)

G71 is for rough turning non-form stock into form workpiece, and can carry out multiple cuts parallel to Z axis according to the specified machining path A—A'—B and depth of cut Δd as shown in Figure II-31. Finally, it will carry out cutting with allowance Δw and Δu/2 for finishing.

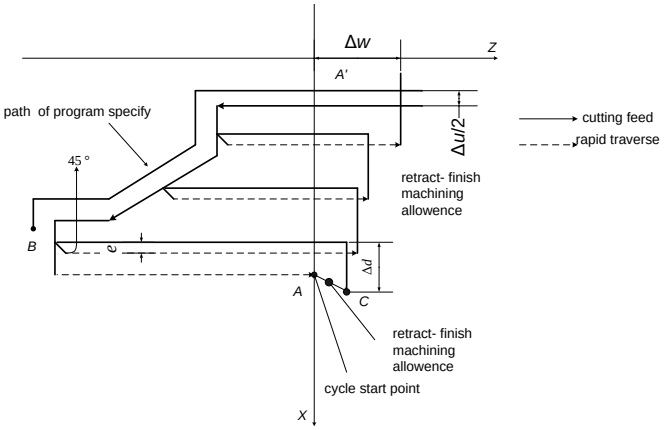


Figure II-31 Rough turning cycle for outer diameter

Command format

G71 U(Δd) R(e) ; (First part)

G71 P(ns) Q(nf) U(Δu) W(Δw) F(f) S(s) T(t) ; (Second part)

N(ns) ;

……

N(nf) ;

Command description

1 △d——Depth of cut (specified by radius, unit: mm), without sign. The cutting direction depends on the AA' direction. This command is modal. Also this value can be specified by the parameter PA21, and the parameter is changed by program command.

2 e——Retraction amount (unit: mm). This value is modal. Also this value can be specified by the parameter PA22, and the parameter is changed by program command.

3 ns——Sequence number of the first block of the program for finishing shape.

4 nf——Sequence number of the end block of the program for finishing shape.

5 △u——Distance and direction of finishing allowance in X direction (specified by diameter, unit: mm).

6 △w——Distance and direction of finishing allowance in Z direction (unit: mm).

7 In the MDI mode, G71 command is disabled.

8 The following commands shall not appear in the range of P and Q specified block.

- Non-modal command except for G04.
- Group 01 command except for G00/G01/G02/G03.
- M98/M99 command.

9 Same sequence number is not allowed in the P and Q specified block.

10 It is possible to insert manual operation during G71 cycle. Before restarting G71 cycle after inserting manual operation, it is necessary to return to the position that is before inserting manual operation, otherwise incorrect position will occur.

11 In the G71 rough machining cycle, only functions F, S, T that are included in G71 are valid. And the functions F, S, T in blocks ns→nf are valid only at finishing but invalid at roughing.

12 Between A' and B, both X and Z axis tool path must conform to simultaneous monotone increase or decrease pattern.

13 When an option of constant surface speed control is selected between ns~nf blocks, G96 or G97 command are invalid at roughing cycle. G96 or G97 command that is included in G71 block or the previous block is valid at roughing cycle.

14 The block for path from A to A' in sequence number "ns" can include G00 or G01, but no Z axis movement command.

15 There are four cutting modes as following for G71. All are executed according to the tool movement parallel to Z axis, and the signs of △ U and △W for finishing allowances are as shown in Figure II-32

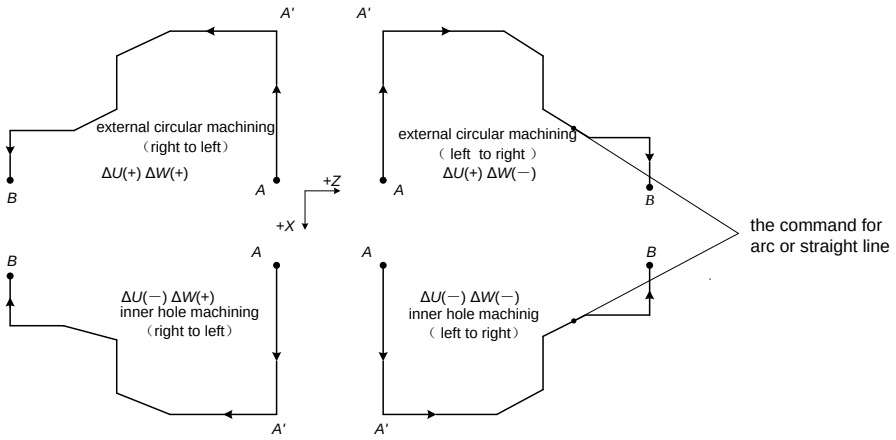


Figure II-32 Signs of Δ u and Δw

3.15 Multiple Repetitive Rough Facing Cycle (G72)

G72 is for rough turning non-form stock into form workpiece, and can carry out multiple cuts parallel to X axis according to the specified machining path A—A'—B and depth of cut Δd as shown in Figure II-33. Finally, it will carry out cutting with allowance ΔW and ΔU/2 for finishing.

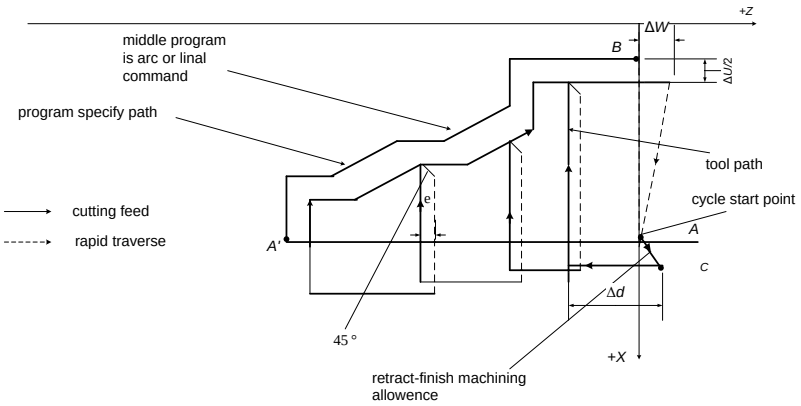


Figure II-33 Rough facing cycle

Command format

G72 U(Δd) R(e); (First part)

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

N(ns) ;

……

N(nf) ;

Command description

1 Δd——Depth of cutting (specified by radius, unit: mm), without sign. The cutting direction depends on the AA' direction. This value is modal. Also this value can be specified by the parameter PA21, and the parameter is changed by program command.

2 e——Retraction amount (unit: mm). This value is modal. Also this value can be specified by the parameter PA22, and the parameter is changed by program command.

3 ns——Sequence number of the first block of the program for finishing shape.

4 nf——Sequence number of the last block of the program for finishing shape.

5 ∆U——Distance and direction of finishing allowance in X direction (specified by diameter, unit: mm).

6 ∆W——Distance and direction of finishing allowance in Z direction (unit: mm).

7 In MDI mode, G72 command is disabled.

8 The following commands shall not appear in the range of P and Q specified block.

- Non-modal command except for G04.
- Group 01 command except for G00/G01/G02/G03
- M98/M99 command

9 Same sequence number is not allowed in the P and Q specified block.

10 It is possible to insert manual operation during G72 cycle. Before restarting G72 cycle after inserting manual operation, it is necessary to return to the position that is before inserting manual operation, otherwise incorrect position will occur.

11 In the G72 rough machining cycle, only functions F, S, T that are included in G72 are valid. And the functions F, S, T in blocks ns→nf are valid only at finishing but invalid at roughing.

12 Between A' and B, both X and Z axis tool path must conform to simultaneous monotone increase or decrease pattern.

13 When an option of constant surface speed control is selected between ns~nf blocks, G96 or G97 command are invalid at roughing cycle. G96 or G97 command that is included in G72 block or the previous block is valid at roughing cycle.

14 The block for path from A to A' in sequence number "ns" can include G00 or G01, but no Z axis movement command.

15 There are four cutting modes as following for G72. All are executed according to the tool movement parallel to Z axis, and the signs of ∆U and ∆W for finishing allowances are as shown in Figure II-34.

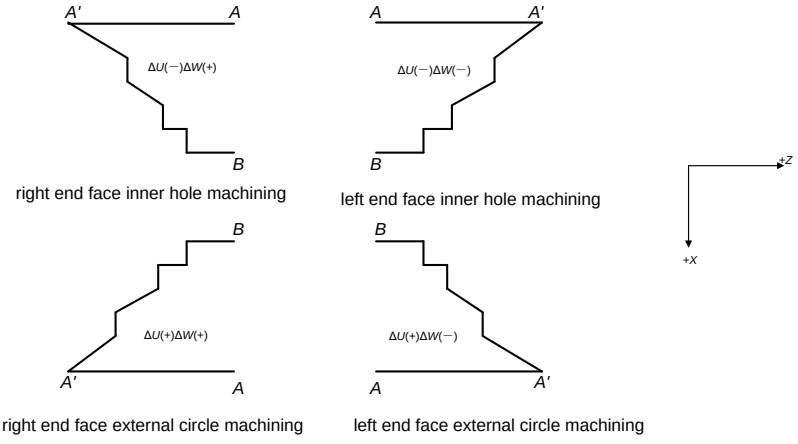


Figure II-34 Signs of ∆ U and ∆W

The Tool path between A and A' is specified in the block with sequence number "ns" including G00 or G01, and in this block ,a move command in the X axis cannot be specified .The tool path between A' and B must be steadily increasing and decreasing pattern in both X and Z axis.

3.16 Multiple Repetitive Turning Cycle (G73)

G73 is for rough turning form stocks. Canned turning cycle can gradually approach the final form according to the cutting form, and the mode is an efficient way for cutting cast or forged stocks. G73 cycle mode is as shown in Figure II-35.

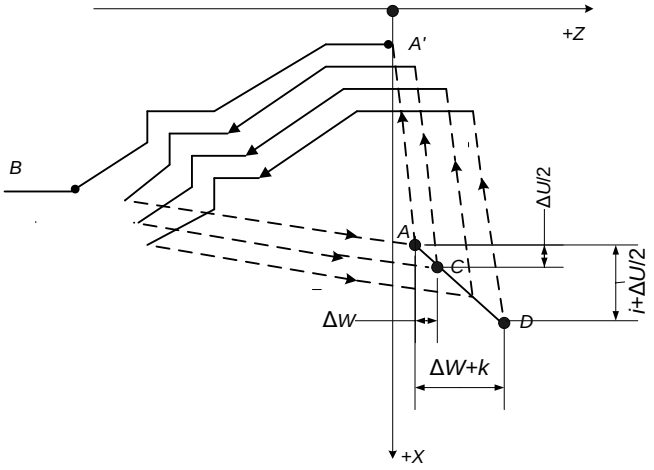


Figure II-35 Canned turning cycle

Command format

G73 U(i) W(k) R(d); (First part)

G73 P(ns) Q(nf) U(Δu) W(Δw) F(f) S(s) T(t) ; (Second part)

N(ns)

……

N(nf)

Command description

1 i——Distance and direction of retraction in the X axis (specified by radius), namely, the radius value with maximal cutting allowance in X axis. This specification is modal. Also this value can be specified by the parameter PA23, and the parameter is changed by program command.

2 k——Distance and direction of retract in the Z axis direction. This specification is modal and is not changed until another value is designated. Also this value can be specified by the parameter PA24, and the parameter is changed by program command.

3 d——The number of division, i.e. the repetitive counts for rough turning. This specification is modal and is not changed until another value is designated. Also this value can be specified by the parameter PA25, and the parameter is changed by program command.

Example: R0.001 indicates the repetitive count for rough turning is 1, R1 indicates the repetitive count for rough cutting is 1000.

4 ns——Sequence number of the first block for finishing shape.

5 nf——Sequence number of the last block for finishing shape.

6 Δu——Distance and direction of finishing allowance in X direction (specified by diameter, unit: mm).

7 Δw——Distance and direction of finishing allowance in Z direction (unit: mm).

8 In MDI mode, G73 command is disabled.

9 The following commands shall not appear in the range of P and Q specified block.
- Non-modal command except for G04.
- Group 01 command except for G00/G01/G02/G03
- M98/M99 command

10 Same sequence number is not allowed in the P and Q specified block.

11 It is possible to insert manual operation during G73 cycle. Before restarting G73 cycle after inserting manual operation, it is necessary to return to the position that is before inserting manual operation, otherwise incorrect position will occur.

12 In the G73 rough machining cycle, only functions F, S, T that are included in G73 are valid. And the functions F, S, T in blocks ns→nf are valid only at finishing but invalid at roughing.

13 Between A and B, both X and Z axis tool path must show simultaneous monotone increase or decrease pattern.

14 When an option of constant surface speed control is selected between ns~nf blocks, G96 or G97 command are invalid at roughing cycle. G96 or G97 command that is included in G73 block or the previous block is valid at roughing cycle.

15 The block for path from A to A' in sequence number "ns" can include G00 or G01, but no Z axis movement command.

16 Cycle motion is carried out according to P~Q program in G73 command. The cutting shape is divided into four types, take care of signs of Δu, Δw, i, k at programming. The tool automatically returns to point A after cycle end.

17 For G73 command, X and Z axis monotone increase/decrease is not applied.

3.17 Multiple Repetitive Finishing Cycle (G70)

G70 command makes it possible to complete finish machining allowance in one pass after G71, G72, G73 rough machining. After G70 cycle terminates, the tool returns to the start point to execute the next block after G70.

Command format

G70 P(ns) Q(nf) ;

Command description

After rough cutting by G71, G72 or G73, the following command carries out finishing.

1 ns——sequence number of the first block for the program of finishing path.

2 nf——sequence number of the last block for the program of finishing path.

3 In MDI mode, G70 command is disabled.

4 The following commands shall not appear in the range of P and Q specified block.
- Non-modal command except for G04.
- Group 01 command except for G00/G01/G02/G03.
- M98/M99 command

5 Same sequence number is not allowed in the P and Q specified block.

6 It is possible to insert manual operation during G70 cycle. Before restarting G70 cycle after inserting manual operation, it is necessary to return to the position that is before inserting manual operation,

otherwise incorrect position will occur.

7 At finishing, F, S and T functions specified in the block G71, G72, G73 are invalid, but those F, S and T functions specified in ns→nf are valid. When the cycle of G70 ends, the tool returns to the start point at rapid feedrate and the next block is read.

Example of G70 and G71

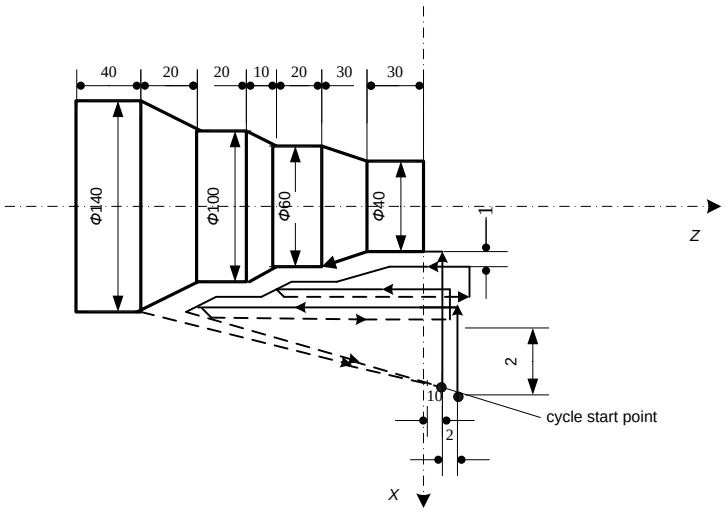


Figure II-36 Use of G70 and G71

```
N10 M03 S××;
N20 T0101;
N30 G00 X160 Z10;
N40 G71 U2 R1;
N50 G71 P60 Q120 U2 W1 F100 S××;
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;          specifying cutting path for finishing
N140 G00 X200 Z50;
N150 T0100 M05;
N160 M30;
```

Example of G70 and G72

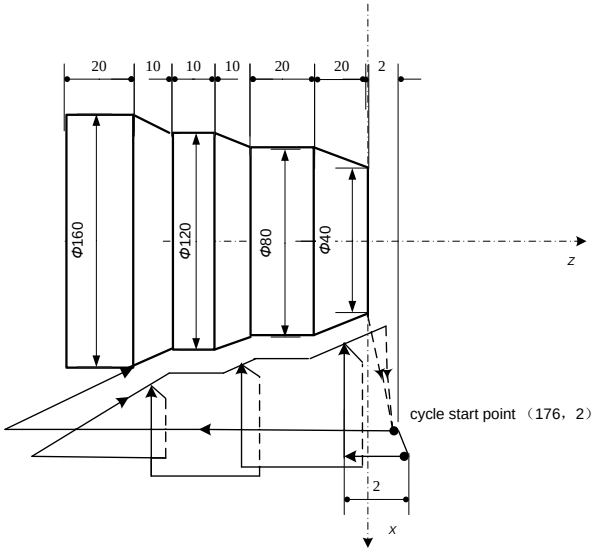


Figure II-37 Use of G70 and G72

```
O0002;
N10 M03 S××;
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;      Specifying the cutting path for finishing
N140 G00 X200 Z50;
N150   T0200 M05;
N160 M30;
```

Example of G70 and G73

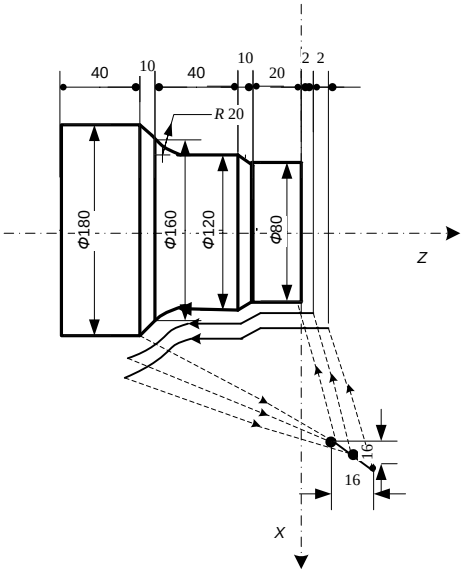


Figure II-38 Use of G70 and G73

```
O0003;
N10 M03 S××;
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;          Specifying machining path for finishing
N140 T0300 M05;
N150 M30;
```

3.18 Multiple Repetitive End Peck Drilling Cycle (G74)

The command allows peck drilling.

Command format

G74 R(e) ;

G74 Z(w) Q(Δk) F(f) ;

Command description

1 e——Retraction amount. This specification is modal. Also this value can be specified by the parameter P'026 and the parameter is changed by program command (unit: mm).

2 Z(w)—— Drilling depth in Z axis, without sign (unit: mm).

3 Δk——Movement amount in Z axis, without sign (unit: μm)

4 F——Cutting feedrate.

5 The tool path is as shown in Figure II-39 as G74 is performed.

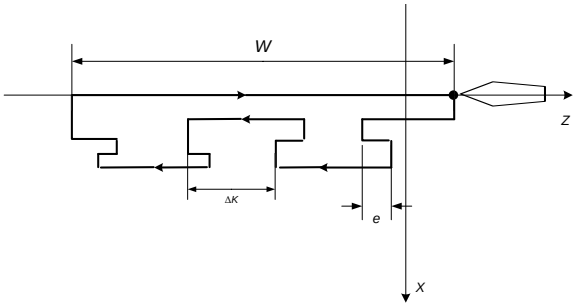


Figure II-39 Cycling process of G74

Example

```
N10 G00 X0 Z10;
N20 G74 R2;
N30 G74 Z-80 Q10000 F800;
N40 G00 X50 Z50;
N50 M30;
```

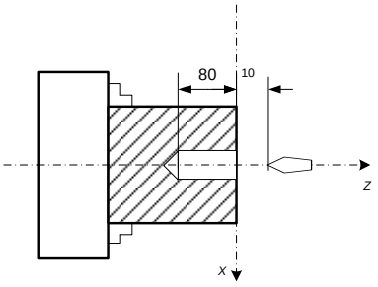


Figure II-40 Use of G74 (peck drilling cycle)

3.19 Multiple Repetitive Outer Diameter Slotting Cycle (G75)

The command allows machining and cutting off of outer diameter slot.

Command format

```
G75 R(e);
G75 X(U)__ P(Δi)__ F(f)__;
```

Command description

1 e——Retraction amount

2 X(U)——Depth of the slot

3 Δi——Depth of cut in X direction (without sign)

4 f—— Feedrate

5 The tool path is as shown in :

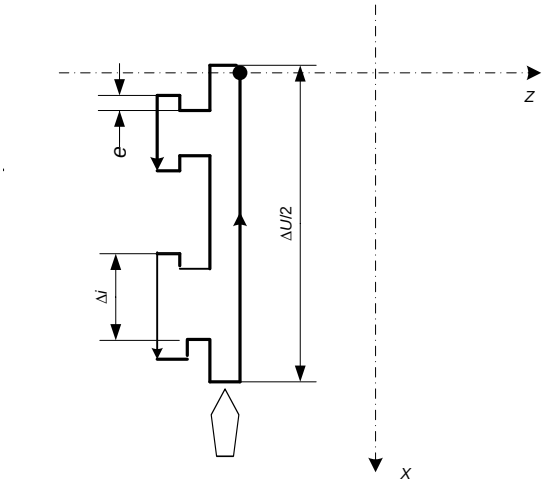


Figure II-41 Use of G75 cycle

Example

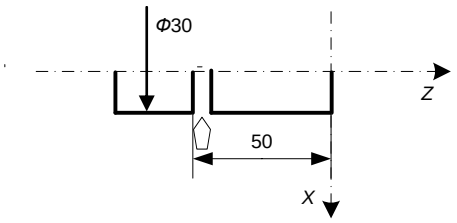


Figure II-42 G75 example

Program:

O0004;

N10 M03 S××;

N20 T0101;

N30 G00 X35 Z-50;

N40 G75 R1; Retracting amount:1mm

N50 G75 X-1 P5000 F60; Cutting several times: 5mm/time

N60 G00 X100 Z50 M09;

N70 M05;

N80 T0100;

N90 M30;

3.20 Constant Surface Speed Control (G96, G97)

A spindle transducer enables the spindle to run at constant surface speed. The constant surface speed control means that the linear velocity following S is constant. With the tool position changes, the spindle speed is calculated according to linear velocity, and the corresponding voltage is output to the spindle controller so that a constant cutting velocity is maintained between the instantaneous position of the tool and the workpiece surface.

The unit of linear velocity is m/min.

G96 command

Constant surface speed controlling is valid, and the cutting linear velocity (m/min) is set.

- **Command format**

  G96 S__ ;

- **Command description**

  G96 is a modal command, linear velocity range: S0000～S9999, preceding zero may be omitted.

G97 Command

Cancels constant linear velocity control command, and sets spindle speed (rpm)

- **Command format**

  G97 S__ ;

- **Command description**

  G97 is a modal command, spindle speed range: S0000~S9999, preceding zero may be omitted.

Limitation of maximum spindle speed

Sets the maximum spindle speed (rpm) at constant linear velocity control;

- **Command format**

  G50 S_ ;

- **Command description:**

  Under constant surface speed control, when the spindle speed is higher than the maximum spindle speed specified by G50, the spindle speed is clamped at the maximum spindle speed.

Notes on constant surface speed control

1 For the rapid traverse block specified by G00, the constant surface speed control is valid only at the end point of G00 command. In the cutting block specified by G01, G02 and G03, etc., the constant surface speed control is valid.

2 The S value specified in G96 is maintained even in G97 and is restored when returning to G96.
   - G96 S50; (setting constant surface speed: 50 m/min)
   - G97 S1000; (canceling constant surface speed: setting speed 1000 rpm)
   - G96 G01 X100; (constant surface speed is valid: 50 m/min)

3 When G96 mode turns into G97 mode, the last spindle speed specified in the G96 command is used as the S value in the G97 mode if S (rpm) is not specified in the G97 block.

  X is the spindle speed in the block before N300. When the G96 mode turns into G97 mode, the spindle speed does not change. When G97 turns into G96, the value of S in the G96 mode is valid. If S is not specified ever, S = 0 m/min.

- N100 G97 S800; (800 rpm)
- N200 G96 S100; (100 m/min)
- N300 G97; (X rpm)

4 Even when a machine that is operating is under machine lock state, the constant surface speed is calculated according to the change in the coordinate of X axis in the program.

5 The constant surface speed control is also valid during thread cutting. Therefore, in order that the spindle will rotate at identical speed, it is recommended that the constant surface speed control be invalidated via G97 command during thread cutting.

6 G99 and G96 can be valid at the same time.

7 The linear velocity specified in constant surface speed control is relative to the program path, namely the tool tip, rather than the linear velocity of the position after tool compensation.

Example

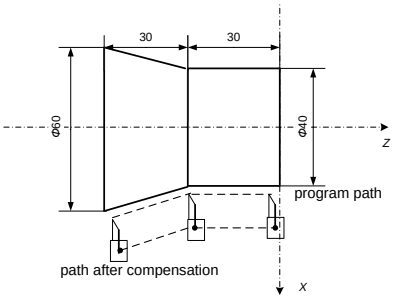


Figure II-43 Example for constant surface speed cutting

```
N10 ;
NG00 X100 Z80;
N40 T0101;
N50 X40 Z10;
N60 G50 S2000;          Specifying the maximum spindle speed
N70 G96 S200;           Linear velocity 200 m/min
N80 G01 Z-30 F100;
N90 X60 Z-60;
N100 Z－65;
N110 G97 S500;
N120 ;
```

3.21 Per-minute/Per-revolution Feed (G98/G99)

G98 command

Specify the cutting feedrate (mm/min)

- **Command format**

G98 F__;

- **Command description**

G98 is a modal command. Speed range: F0001~F8000, the preceding zero can be omitted.

G99 command

Specify the cutting feedrate (mm/rev)

- **Command format**

G99 F__;

- **Command description:**

G99 is a modal command. Speed range: F0.0001~F0500, the preceding zero can be omitted.

3. Remarks on G98/G99 feed commands

1 G98 and G99 are modal G commands of the same group, and only one from them will be valid at same time. When the system is power ON, G98 is default mode.

2 The conversion formula between the feed per-revolution (Fr) and the feed per -minute (Fm) is:
Fm = Fr×S 【S: Spindle speed(rpm)】

3 When G99 modal cutting is to be done, the machine must be equipped with a spindle encoder.

3.22 Tool Nose Radius Compensation (G40~G42)

The command makes it possible to compensate the tool nose radius and improve the machining precision

Command fomat

G40/G41/G42 G00/G01 X Z T ;

Command description

As for tool compensation C function in details, please refer to section 2.7.3

4 FEED FUNCTION (F CODE)

Command format

F××;

Command function

Feedrate of linear interpolation (G01), circular interpolation (G02, G03), etc. are specified with F code and numbers following it.

Command description

1 Cutting feed function controls the tool to feed along the tangential direction at specified F value. The top limitation of cutting feed rate is specified by PA25, if the actual cutting feedrate exceeds the top limitation, it is clamped on the top limitation. The top limitation unit is +mm/min. In the Position page, F feed rate can be adjusted by selecting 0~150％ (step: 10％) override via 

or 

on the operator's panel.

2 There are two ways for expressing feed rate F: Per-minute Feed (G98) and Per–revolution Feed (G99)

- **Per–minute feed (G98), unit is mm/min**

G98 G01 X50 Z50 F100;

Above block indicates that the tool moves to the position (X50 Z50) at cutting feed rate 100mm/min. G98 is a model code. After power ON, the per-minute feed mode is automatically valid. KND system default mode is G98.

- **Per–revolution feed (G99), unit is mm/rev**

G99 G01 X50 Z50 F0.2;

Above block indicates that the tool moves to the position (X50 Z50) at cutting feed rate 0.2mm/rev. G99 is a model code. System default G98 mode unless G99 is specified.

NOTE

1 F code allows max entry 7 bits, if the feed rate exceeds limitation, movement will be clamped at the limit value rather than the specified value.

2 2. In the mode of G99, the spindle must be equipped with a position encoder (1024 line)

5 MISCELLANEOUS FUNCTION

M function consists of M and the 1-2 digits followed, is for controlling the On/Off of the output interface of the machine tool.

Command format

M × ×;

Command description

1 When a movement command and miscellaneous function are specified in the same block, the commands are performed simultaneously.

2 In one block, only one M code will be valid.

5.1 Program Stop (M00)

Command format

M 00(or M0);

Command function

M00 stops execution of the current program and keep the curser on the block M00 locates. Press [START] to restart the execution again.

5.2 Program End (M02)

Command format

M 02 (or M2);

Command function

When executing a block including an M02 which is not the last block of the whole program, system processes all the commands in the block and stops. If the executing block including M02 is the last one of the program, after executing, system locates cursor at the block. If you want to execute the program again, move the cursor to the start position first.

5.3 Program End (M30)

Command format

M 30 ;

Command function

1 In AUTO mode, M30 makes it possible to terminate current block after current block's other

commands are executed, and AUTO mode ends. The workpiece counter increases by 1.

2 By setting BIT0 of parameter PA43 to 1, the cursor will not return to the program start; by setting BIT0 of parameter PA43 to 0, the cursor returns to the program start immediately after the program ends.

3 After executing M30 command, the outputting of signal M03, M04 and M08 etc. are turned off.

5.4 Spindle Control (M03/M04/M05)

Command format

M03 (or M3) ; Make spindle rotate in the CW direction

M04 (or M4) ; Make spindle rotate in the CCW direction

M05 (or M5); Stop spindle rotating

5.5 Coolant On/Off (M08/M09)

Command format

M08/M09 ;

Command function

Control the coolant of machines

5.6 Lube On/Off (M32/M33)

Command format

M32/M33 ;

Command function

Control the lubrication of machines

5.7 Chuck Clamp/Unclamp (M10/M11)

Function parameter

The chuck controlling function is valid when QPSL is set to 1.

036				QPSL				

4 QPSL Select the chuck function

0: Invalid

1: Valid

Relative parameters

041	QPLS	QPM3						

7 QPLS Select the output signal type of chuck

0: Level signal

1: Pulse signal. The pulse width is specified by P51

6 QPM3 Weather check the chuck state when starting the spindle

0: Check the state. If the chuck is loose, system issues an alarm and stops execution.

1: Don't check the state. Ensure that the chuck state is logic high when QPIN is set to 1.

043	QPIN							

7 QPIN Weather the chuck state is detected by some input signals

0: No detecting signals

1: Have detecting signals. When starting spindle, these input signals are detected.

051	QPLSTIME: Width of chuck controlling pulse signal

[Range] -32768~32767

[Unit] ms

[Def. value] 32

Inner/outer chuck selection

In the DEBUG page, you can select inner/outer chuck function by pressing the number key "0"; the differences between inner and outer chuck are as follows:

1) The meanings of the DIAGNOSIS 005 signal QPJ and DIAGNOSIS 005 signal QPS are opposite;

2) The output point and input in-position signals are reverse during QPJ (Chuck Clamped) testing.

Chuck control command (M10/M11)

M10 : Chuck clamp

M11 : Chuck unclamp

NOTE

Output point of M code is different for internal/external chuck selection.

DI/DO signal

DGN.0				QPI				

4 QPI Input signal form foot switch for chuck

DGN.3	QPJI	QPSI						

7 QPJI Indicate weather inner chuck clamping or not

0: Not clamping

1: Clamp in-position

6 QPSI Indicate weather outer chuck clamping or not

0: Not clamping

1: Clamp in-position

DGN.5	QPJ	QPS						

7 QPJ Output signal for clamping chuck

6 QPS Output signal for unclamping chuck

The motion sequence of the chuck is shown in Figure II-44 (Level control):

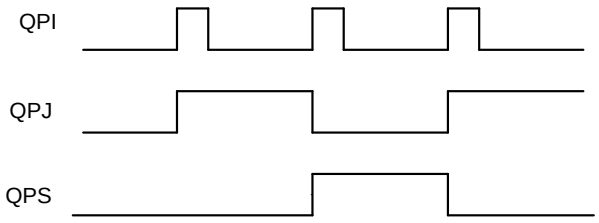


Figure II-44

NOTE

1 The signal QPJ and QPS are all zero at power ON.

2 Setting QPM3 to 0, the chuck must clamp when spindle starts; otherwise alarm (No.015) will occur: Spindle Started with Unclamped Chuck.

3 The QPI signal is invalid when the spindle is rotating and when the spindle is braking.

5.8 Tailstock Forward/Backward (M78/M79)

M78 and M79 are used to control tailstock.

Parameters

When the position parameter TWSL of PA41 parameter is set to 1, the tailstock function is valid.

041					TWSL			

3 TWSL Select the tail function

0: Invalid

1: Valid

I/O signals

- **Input signal**

DGN.3						TWI		

2 TWI Input signal form foot switch for tail

- **Output signal**

DGN.5			TWJ	TWT				

5 TWJ Output signal for driving tailstock move forward

4 TWT Output signal for driving tailstock move backward

The time sequence diagram for the tailstock movement is shown in Figure II-45:

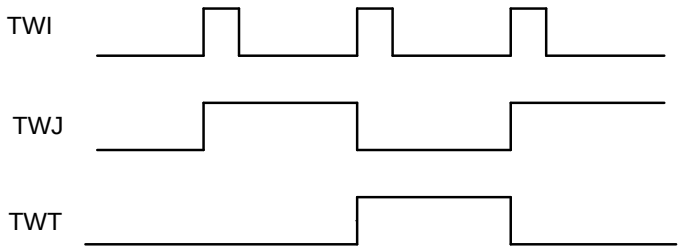


Figure II-45

NOTE

The I/O points used by the tailstock controlling are same as those for M21 and M23. Therefore, the parameter of TWSL and M210, M230 shall not be set to 1 at the same time. Otherwise, the system will automatically set them as 0.

Automatic control (M code control)

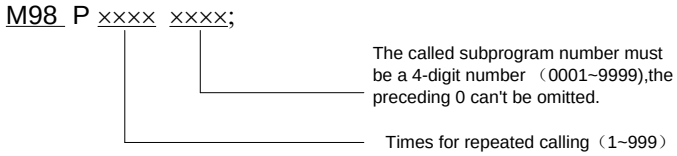
M78: the system outputs tailstock forward signal (TWJ=1)

M79: the system outputs tailstock backward signal (TWT=1)

5.9 Call Subprogram (M98)

In Auto mode, when executing M98 command, the system will call the subprogram specified by P after other commands in the current block have been performed.

Command format



Command description:

If the times number for repeated calling is omitted, the default times for repeated calling is 1. The command is Invalid in the MDI mode.

5.10 Return from Subprogram (M99)

Command format

M99 ;

Command function

Return from subprogram.

5.11 Special M codes (M21~M24)

M21/M22 code

Format 1 : M21;

Function 1: If M21O of No. 041 parameter is 1, M21O will be output.

Format 2 : M22;

Function 2: If M21O of No. 041 parameter is 1, M21O output will be turned off.

Format 3 : M21 P_ ;

Function 3: To end after executing a time specified by P_. If M21O of No. 041 parameter is 1, M21O

will be output. At ending, the output M21O is turned off. Unit of P: ms.

Format 4 : M21 Q_ ;

Function 4: To detect the input port M21I, and end when an input signal is available. If M21O of No. 041 parameter is 1, M21O will be output. At ending, the output M21O is turned off. Q can be specified in any way.

M23/M24 code

Format 1 : M23;

Function 1: The execution is identical with normal M code. If M23O of No. 041 parameter is 1, M23O will be output.

Format 2 : M24;

Function 2: If M23O of No. 041 parameter is 1, M23O output will be turned off.

Format 3 : M23 P_ ;

Function 3: To end after executing a time specified by P_. If M23O of No. 041 parameter is 1, M23O will be output. At ending, the output M23O is turned off. Unit of P: ms.

Format 4 : M23 Q_ ;

Function 4: to detect the input port M23I and end when an input signal is available. If M23O of No. 041 parameter is 1, M23O will be output. At ending, the output M23O is turned off. Q can be randomly specified.

I/O signals

DGN.3							M23I	M21I
DGN.5			M23O	M21O				

5.12 User Interface Skip Function M Codes (M91~M94)

Command format

M9* Pn ; (* is 1or 2 or 3or 4)

Command function

M91P_; : Go to the specified block when the input interface M91I is 0; execute in sequence when it is 1.

M92P_; : Go to the specified block when the input interface M91I is 1; execute in sequence when it is 0.

M93P_; : Go to the specified block when the input interface M93I is 0; execute in sequence when it is 1.

M94P_; : Go to the specified block when the input interface M93I is 1; execute in sequence when it is 0.

Command description

go to the block specified by n when the condition is satisfied; ALARM (№148) occurs when n is not detected. Alarm (№76) occurs if P is not programmed.

I/O signals

DGN.3					M93I	M91I		

5.13 Automatic Spindle Gear Shifting (M41~M42)

Command format

M4* ; (* is 1 or 2)

Command function

It is used to shift gear of analog spindle automatically.

Command description

Refer to section 2.6.3 for details.

5.14 Automatic Lubrication Function

Function description

When the parameter P043.1 (AURH) is set to 1, the automatic lubrication function is enabled. When the system executes M32 command, it turns on the output of signal. When M32 command is not specified, the M32 port outputs automatically as preset. The time for turning on/off the output is specified by parameters.

052	Time of automatic lubrication-ON

[Range] 1～65535

[Unit] ms

[Def. value] 16

053	Time of automatic lubrication-OFF

[Range] 1～65535

[Unit] ms

[Def. value] 16

Alarms

If parameter P054.2 (RHLOW) is set to 1, the “alarm for too low lubricant level” is enabled. The input port for this function is diagnosis 003.5 (X35):

DGN.3			X35					
Interface			X54:3					

When diagnosis 003.5＝1, it is normal; when 003.5＝0, “alarm for too low lubricant level” will be given off.

NOTE

1 During Auto mode, if this alarm occurs, the machine movement will not stop.

2 If the alarm occurs when the machine is not under movement, the machine can’t be started until the alarm is removed.

6 SPINDLE FUNCTIONS (S CODE)

Code signals are sent to a machine through address S and the values following S to control the spindle speed. One S code can be set in a block. When a movement command and a S code are in the same block, they will be performed at the same time. There are two ways for controlling spindle speed: one is stepped speed-variation controlling, another is staples speed-variation controlling (with spindle transducer).

6.1 Step Spindle Speed Variation (Digital Controlling)

Command format

S××;

Command function

The spindle's speed is controlled by command consisting of S and 2-digit, which can adjust several stepped speed of a spindle.

Command description

1 To achieve stepped speed-variation controlling for a spindle, SANG of parameter P004 must be cleared to 0.

2 The output interface of XS57 of K100Ti-B can directly output 4 steps of spindle speed-variation signals, i.e. S01, S02, S03 and S04, that is, 4 steps of spindle-speed controlling can be achieved. The output of S01~S04 is turned off when executing S00.

Relative parameters

045	STIME1: Delay time 1 for S code when changing gear
	[Range] 0～4080
	[Unit] ms
	[Def. value] 16
046	STIME2: Delay time 2 for S code when changing gear
	[Range] 0～4080
	[Unit] ms
	[Def. value] 16

STIME1: A time 1: 0~4080 ms is delayed when the spindle gear shifts via S code.

STIME2: A time 2: 0~4080 ms is delayed when the spindle gear shifts via S code.

For example, when changing spindle speed from S1 to S2, system firstly turns of the S1 output signal, after a delay time specified by STIM1, it turns on the S2 output signal. After a delay time specified by STIME2, system executes the next block.

STIME1 delay is omitted when changing spindle speed from S0 to S* or from S* to S00.

6.2 Step less Spindle Speed Variation (Analog Controlling)

Command format

S××××;

Command function

The spindle speed (rpm) is directly controlled by address S+ four digits, which can achieve step less speed variation for a spindle.

Command description

1 To achieve steeples speed-variation controlling, SANG should be set to 1.

2 The preceding zero for the four digit of S code can be omitted.

3 At step less speed variation, the spindle speed is controlled by the output voltage of the analog interface of the spindle. When the analog interface of the spindle outputs 10 V, the corresponding spindle speed is the max. speed, that is, the max. speed set in PA31. The corresponding relationship is: Output voltage of the analog interface of the spindle = specified spindle speed S×10V / set value of PA31 parameter.

Example

M03 S500 ; means the spindle starts CW at 500 rpm.

The maximum spindle speed set points are different with respect to different machines. The max. speed is set in PA31. The maximum spindle speed is defined just by writing a known max. spindle speed into the parameter PA31. If a commanded speed in a program exceeds the value, the speed will be limited to the max. speed set in the parameter.

6.3 Automatic Analog Spindle Gear Shifting

Command format

M41 P_ Q_ L_ ;

M42 P_ Q_ L_ ;

Command function

M41 command is used to change spindle gear 1, and M42 to change spindle gear 2.

Command description:

P——delay 1 at gear shifting, unit: ms. It is a modal value, and will maintain after power OFF.

Q——delay 2 at gear shifting, unit: ms. It is a modal value, and will maintain after power OFF.

L——spindle speed at gear shifting, unit: rpm. It is a modal value, and will maintain after power OFF.

Relative parameters

036			AGER					

5 AGER Weather the function of automatic changing spindle gear is valid or not

0: Invalid

1: Valid

Gear-shifting process

The program executes M41 (or M42):

1 Check whether AGER of the parameter PA36 is 1, and analog spindle function is selected (SANG=1 for parameter 004), otherwise ALARM (№01) occurs: M code error.

2 Whether the gear is same as the current gear (the output state is checked), if yes, M code ends, and no gear shifting is conducted. If not, gear shifting process next step 3) will be done.

3 The spindle speed is made to be the speed specified by L. The movement, if any, is paused.

4 After delaying a time set by P, the output signal from the former gear is turned off, and the signal of the new gear is output.

5 The in-place input signal of the gear is checked, if yes, go to next step6)). Otherwise, wait.

6 After delaying a time specified by Q, the analog spindle command value is output according to the new spindle speed, and the gear-shifting M command ends.

I/O signals

DGN.3			M42I	M41I				
DGN.5							M42O	M42O

6.4 Manual Analog Spindle Gear Shifting

Parameter

043			JGER					

5 JGER Select the function of manually changing analog spindle's gear

0: Invalid

1: Valid. The input signal M42I is used to select gears.

Function description

1 When the function for manual analog spindle gear shifting is valid, the input signal M42I＝0, gear 1 is selected, and the system outputs by taking the max. analog spindle speed specified by P031 as the reference. The input signal M32I=1, gear 2 is selected, and the system outputs by taking the maximum analog spindle speed specified by P032 as the reference.

2 When the automatic spindle gear-shifting function is selected (AGER=1), the function is invalid, and the parameter JGER is automatically set to 0.

I/O signals

DGN.3			M42I					

NOTE

Signal X35 has some other functions (e.g. override or T11 tool position) as well, pay attention to avoid any conflict in using them.

6.5 Switch between Step and step less changing speed

Setting analog spindle function enable（parameter P004 SANG=1）and setting parameter P043 SANG2 ＝1, then system can automatically switchover between spindle stepped and step less speed variation , control process show as follows:

1 As performing code S1~S4, corresponding numerical interface（S01~S04）output spindle gear signal, and turn off analog voltage output.

2 As performing code more than S4 , numerical interface output become analog voltage output, and all S code numeral interface output signal turn off.

3 Gear shift sequence show as follows:

As switchover between S1~S4, shift gear sequence is identical to machine gear sequence, refer to section 2.6.1.

From S1~S4 switchover to code more than S4 , turn off gear numeral control signal（refer to Diagnosis 005: S4~S1）, at same time output analog voltage.

Code more than S4 switchover to S1~S4, turn off analog voltage output; firstly, output gear control signal after delay STIMER1 ;secondly, perform next block after delay STIMER2.

Parameter

043								

2 SANG2 Select the function of exchanging state between the step less spindle speed adjusting and the step spindle speed adjusting (It's only available when analog spindle is enabled.)

0: All S commands are treated as step less spindle speed adjusting commands

1: S1~S4 are treated as step spindle speed adjusting commands. Other S commands are treated as step less spindle speed adjusting commands.

7 TOOL FUNCTIONS (T CODE)

7.1 Tool Change

A cutting tool on a machine is selected by address T plus a 4–digit number. When a movement command and a T code are specified in the same block, they will be performed simultaneously.

First two digits of T code indicate the desired tool. Last two digits are used as the offset number indicating the compensation amount for tool offset.

The tool number that the system can supply is specified by P39, the maximum value is 8.

Tool change process

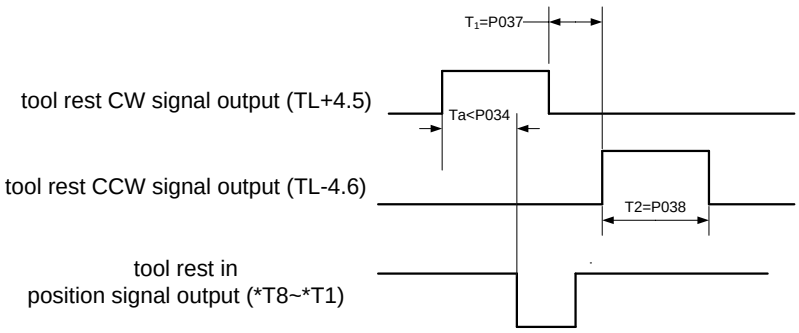


Figure II-46 Tool change process

NOTE

When Ta≥P034, system issues an alarm which indicates that the time of tool change is too long.

When executing T code, first the tool rest CW signal (TL+) is output to rotate the tool rest. The tool rest CW signal is off after receiving the in-position signal specified by T code. After delaying a period specified by parameter PA37, the tool rest rotates CCW and is locked (TL-) for a time width set by P038. Then the CCW signal is off, and tool change process is finished. The next block is performed. If the specified tool number is same as the current one, the tool change command ends at once, and the next block is performed.

Parameters

1 Tool in-position signal (*T8~*T1): TSGN is used to select the active level of these signals.

2 T1: Delay time from tool rest stops rotating in positive direction to starts rotating in negative direction to clamp (P037)

3 T2 : Delay time of outputting clamping signal in negative direction (P038)

4 Tool number (P039)

5 Ta : Maximum delay time when performing the tool exchanging operation (P034)

Alarm

- **ALARM (№03): T code error**

  When the tool number specified by T code exceeds the maximum specified by PA39, the ALARM (№03) will occur, tool change and machining program will stop.

- **ALARM (№05): too long tool change time**

  If the tool change time exceeds the PA34 specified maximum tool change time, the ALARM (№05) will occur, tool change and machining program will stop.

> **NOTE**
>
> About the relationship between the commanded tool number and the actual tool, please refer to the specification of the machine builder.

Check the state of tool rests regularly

When CKTDI is set to 1, system detects the state of tool rest in a fixed period.

1 After tool change, checking the tool rest signal again, if the signal is right, tool change ends; otherwise alarm will occur. Program executing will pause. (Pause signal is generated)

2 Checking the tool rest signal, and judging whether it is same as the system recorded signal.

Alarm (No.08) is issued when some abnormal signal is detected.

> **NOTE**
>
> 1 Check the number of corresponding input signal according to the number of tools specified by parameter PA39.
>
> 2 2. If it is not necessary to check or if a gang tool is used, set CKTDI=0.

Back tool rest selection

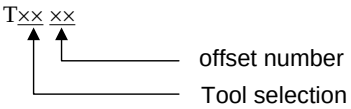
If the machine tool is equipped with a back tool rest, set RVX to 1.

> **NOTE**
>
> When RVX is set to 1, the direction of X+ and X- are reversed.

7.2 Tool Offset

Tool offset is also tool length compensation, which can be simply programmed and operated by compensating the difference between the actually used tool and the assumed tool in programming.

Executing T code can not only carry out tool change but also tool offsetting.



Meaning of T code:

1 Tool selection: Tool selection means corresponding tool number selected on the tool rest.

2 Offset number: It is used to select the offset value. The offset value must be specified by tool offset number at tool compensation page in advance. Every offset number has two offset values, one is for X-axis (diameter), and the other is for Z-axis.

Example T0102

It means to select tool number 1 and execute tool compensation value specified by tool offset number 02. In general, tool no. is same as offset no..

When T code is specified and its offset number isn't 00, offset is valid. If its offset is 00, tool compensation is canceled.

Example T0100

T0100 means that tool #1 is selected and no compensation is performed.

The range of offset value is:－999.999~999.999 mm; ALARM (№29) will occur if out of the range.

NOTE

1 Separate T code: When only a T code alone is specified in a block, the tool does not carry out offset movement until a movement command is specified in the following block. The movement command is performed at rapid traverse rate in G00 mode and at feedrate in other mode.

2 G50 X(x) Z(z) T ; Tool is not offset. The command specifies tool position (x, z) as coordinates system. The tool position is obtained by subtracting the offset value corresponding to the offset number specified in the T code.

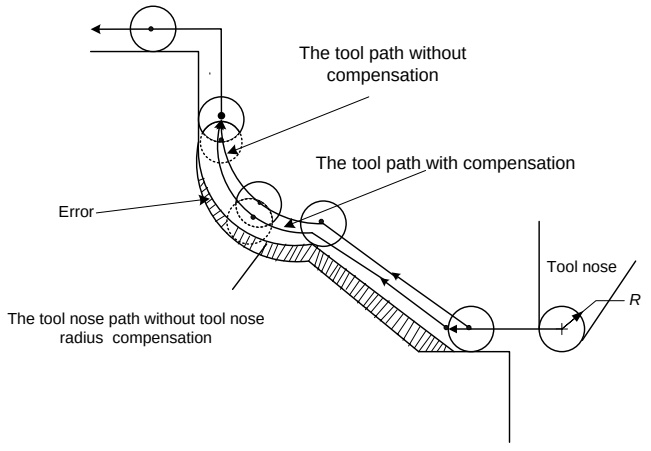
3 Before a program ends, the tool offset can either be cancelled or held, which will not influence the machining; but program end position is different by the tool offset.

4 When only a T code alone is specified in a block, the tool will not be moved at tool offsetting, but the offset value will be subtracted from the absolute coordinates in the Position page of CNC. However the tool can be moved by writing "U 0 W 0" after T××××. When the tool compensation is cancelled, the machine can be moved by writing U0 and W0 in addition to T××00.

5 If a machine tool machines work pieces via gang tools rather than a rotary tool rest, all the tools are regarded as T01 with different tool offset values. For example, T0101 and T0102 may be programmed for two tools. As long as the compensation is specified correctly for each tool, it is possible to call correctly corresponding tool during the machining.

7.3 Tool Nose Radius Compensation (G40~G42)

If a tool nose is circular, it is difficult to write a correct machining program only by tool offset function. The tool nose radius compensation function can automatically compensate above errors.



Command function

Compensate the machining tool nose radius to improve machining accuracy.

Command format

G40/G41/G42 G00/G01 X Z T ;

Command description

G40: canceling tool nose radius compensation

G41: specifying left tool compensation (back tool rest system)

G42: specifying right tool compensation (back tool rest system)

G00/G01: movement command

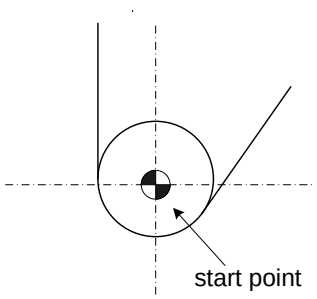
X_/Z_: movement command coordinates

T_ : tool nose direction.

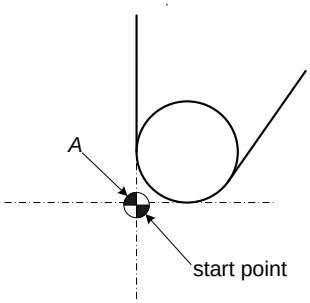
7.3.1 Imaginary tool nose

The tool nose at position A (shown in Fig2-49) does not actually exist. The imaginary tool is necessary because it is usually more difficult to set the actual tool nose center on the start point than the imaginary tool nose. In addition, when the imaginary tool nose is used, the tool nose radius need not be considered in programming.

The position relationship when the tool is set to the start point is shown in below figures.



When programmed using the tool nose center



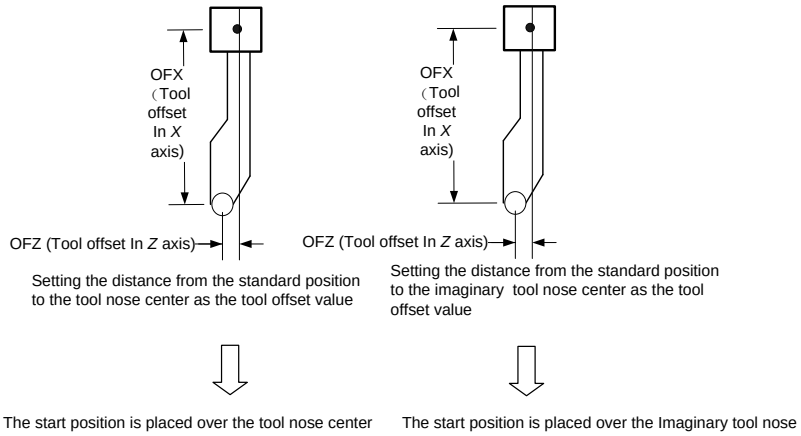
When programmed using the imagingary tool

NOTE

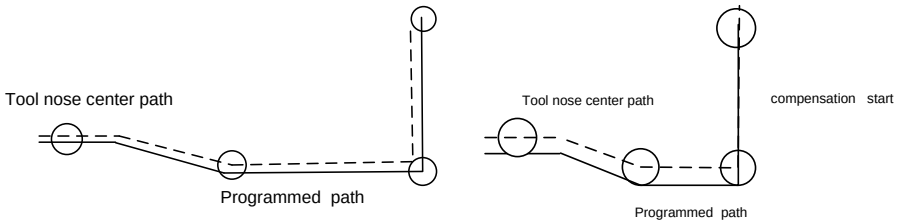
1 As for machine tools with reference points, a standard point like tool rest center can be regarded as a start point. The distance between the standard point and the tool nose radius center or the imaginary tool nose is set as the tool offset value.

2 Setting the distance from the standard point to the tool nose radius center as the offset value is identical with setting the tool nose radius center as the start point, and setting the distance from the standard point to the imaginary tool nose is identical with setting the imaginary tool nose as start point. To set tool offset value, it is usually easier to measure the distance from the standard point to the imaginary tool nose than measure the distance from the standard point to the tool nose radius center.

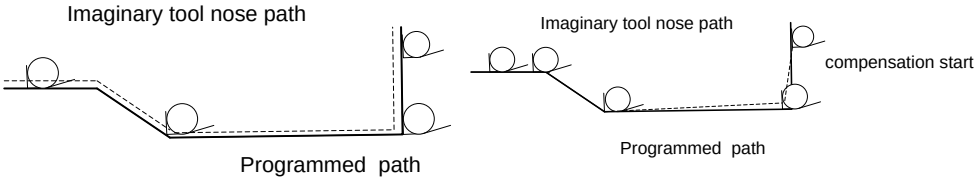
When the tool rest is placed at start point



Programming via the tool nose center

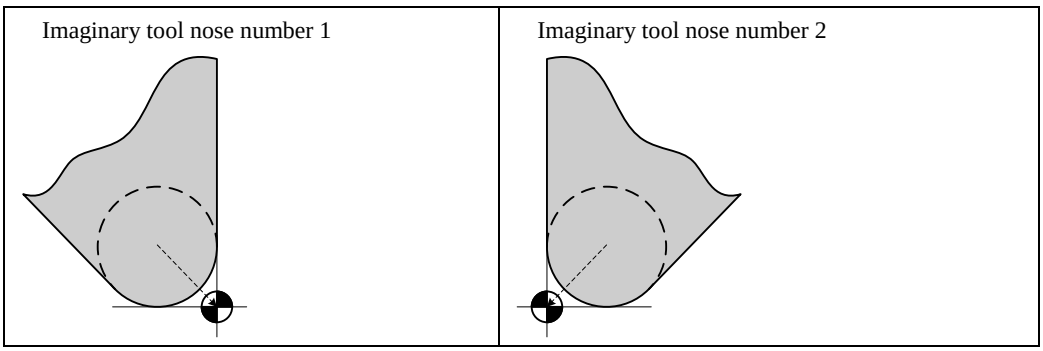


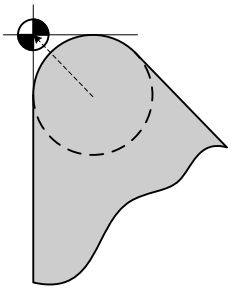
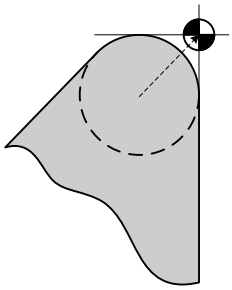
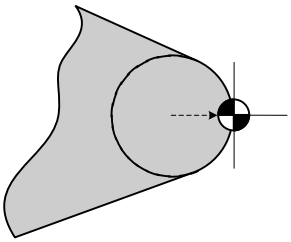
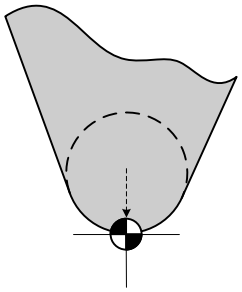
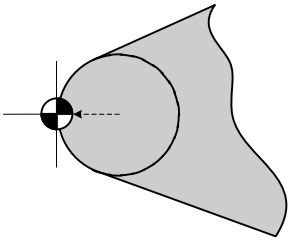
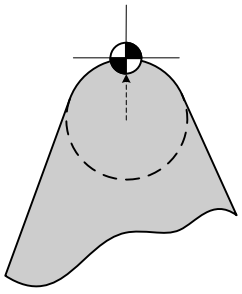
Programming via the imaginary tool nose



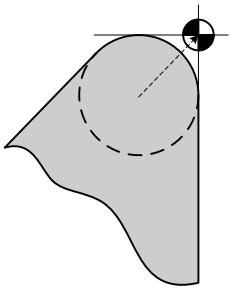
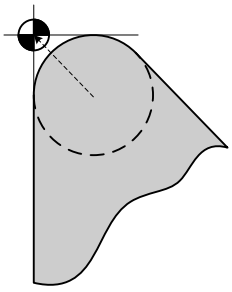
7.3.2 Direction of imaginary tool nose

Front tool rest system

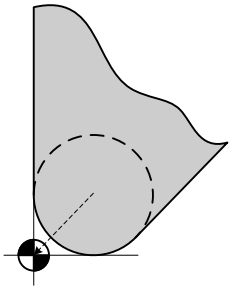


Imaginary tool nose number 3 	Imaginary tool nose number 4 
Imaginary tool nose number 5 	Imaginary tool nose number 6 
Imaginary tool nose number 7 	Imaginary tool nose number 8 

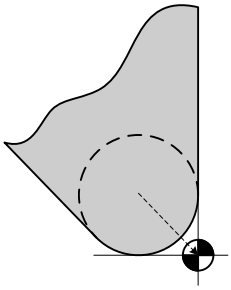
Back tool rest system

Imaginary tool nose number 1 	Imaginary tool nose number 2 

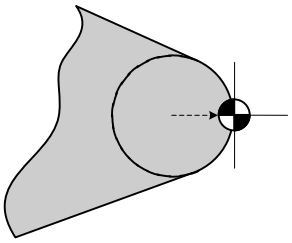
Imaginary tool nose number 3



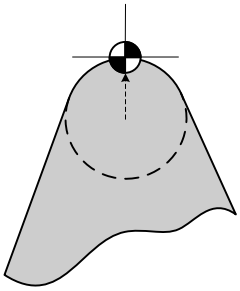
Imaginary tool nose number 4



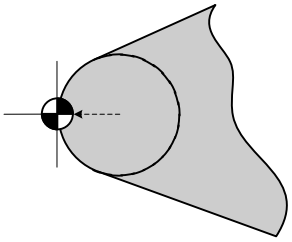
Imaginary tool nose number 5



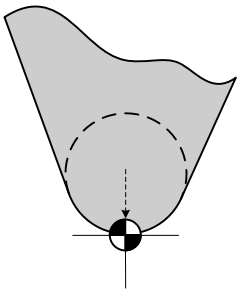
Imaginary tool nose number 6



Imaginary tool nose number 7

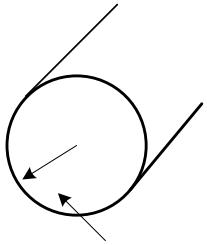


Imaginary tool nose number 8



The direction of the imaginary tool nose relative to the tool nose center is determined by the direction of the tool during cutting, so direction of the imaginary tool nose as well as the offset values must be specified in advance. The direction of the imaginary tool nose can be selected from eight specifications (as shown in above tables). This figure illustrates the relationship between the tool and the start point. End of arrow is the imaginary tool nose. The same tool nose direction number has different tool nose directions in different tool rests (front tool rest or back tool rest).

Set imaginary tool nose number 0 or 9 corresponding to tool compensation number when the tool nose center is identical with the start point. Set imaginary tool nose number via address T.



Tool nose radius compensation value

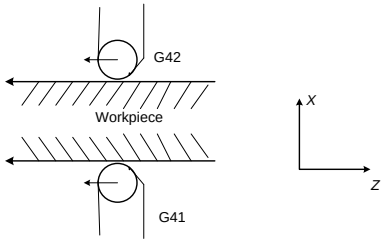
The tool nose radius compensation value is set in MDI mode according to the offset number.

Offset number	Offset of X-axis	Offset of Z-axis	Tool nose radius offset	Direction of imaginary tool nose
001	0.020	0.030	0.020	2
002	0.060	0.060	0.015	3
003	0.030	0.026	0.018	8
……	……	……	……	……
008	0.050	0.038	0.028	1

7.3.3 Work position and movement command

In tool nose radius compensation, the position of the workpiece with respect to the tool must be specified. (Back tool rest system)

G code	Work position	Tool path
G40	Cancel	Moving along the programming path
G41	Right side	Moves along the left side of the programming path
G42	Left side	Moves along the right side of the programming path



Imaginary tool nose on path of parogram center of tool nose on the path of parogram

Imaginary tool nose number 1~8 Imaginary tool nose number 0

The workpiece position can be changed by setting the coordinates system as shown in Figure II-47 (front tool rest system):

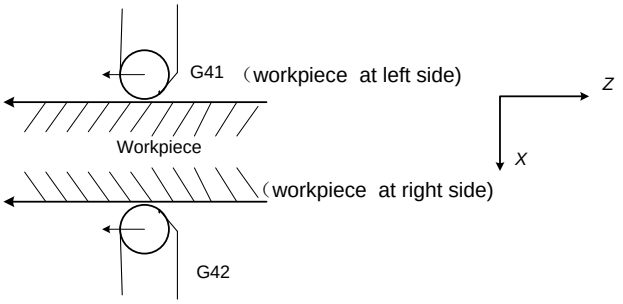


Figure II-47

NOTE

1 If the tool nose radius compensation value is negative, the workpiece position is changed.

2 G40, G41 and G42 are modal.

3 Don't set G41 in the G41 mode, otherwise, compensation will not work normally. Don't set G42 in the G42 mode for the same reason.

When the workpiece position does not change

When the tool moves, the tool nose maintains contact with the workpiece (as shown in Figure II-48).

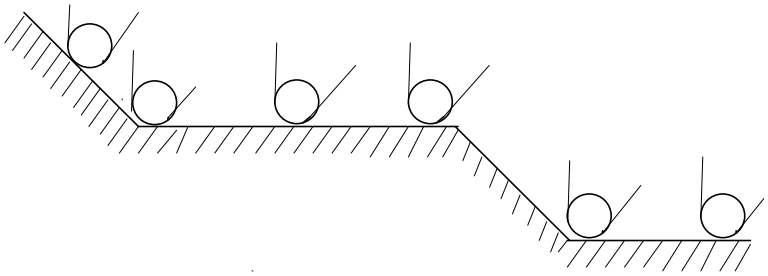


Figure II-48

When the workpiece position changes

At the corner of the programmed path, the workpiece position relative to the tool position changes as shown in Figure II-49.

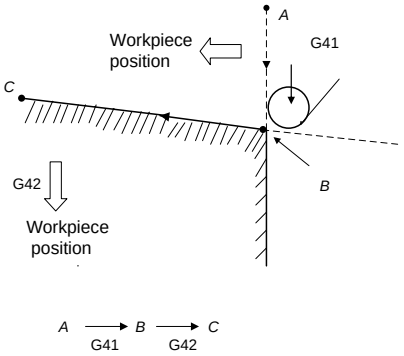


Figure II-49

Although there is no workpiece at right side of the programming path in Figure II-49, the existence of the workpiece is assumed in the movement from A to B. As the workpiece position can't be changed in the block next to tool nose compensation block, in Figure II-49, if the block for moving from A to B is the block for starting tool nose compensation, the tool path would not be the same as Figure II-47.

Compensation start

The block that changes from G40 to G41 or G42 mode is called the compensation start block.

G40 __ ;

G41 __ ; (compensation start block)

__ ;

Transient tool movements for offset are performed in the compensation start block. At the start of the

block after the compensation start block, the tool nose center is vertically to the programmed path.

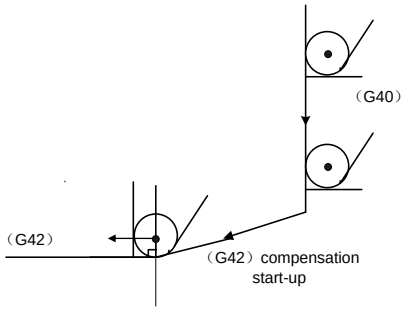


Figure II-50

Offset cancel

Block that changes from G41 or G42 to G40 mode is called the Offset Cancel block.

G41 __ ;

__ ;

G40 __ ; Offset cancel block

__ ;

At the end of previous Offset Cancel block, the tool nose center moves to a position vertical to the programmed path.

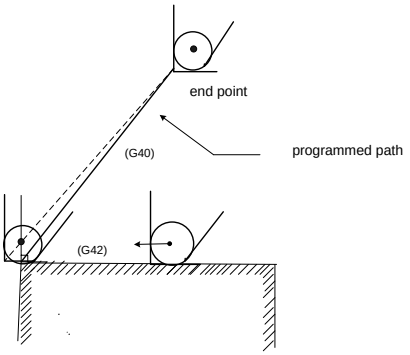
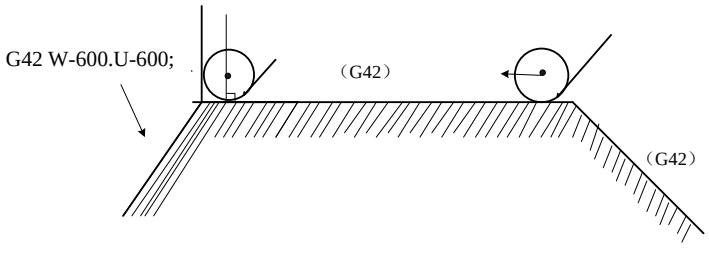


Figure II-51

When G41/G42 is specified again in G41/G42 mode

In this case, the tool nose center, at the end point of the preceding block, is positioned vertical to the previous block programmed path.



In the block that first specifies G41/G42, the above positioning of the tool nose center is not performed.

Example

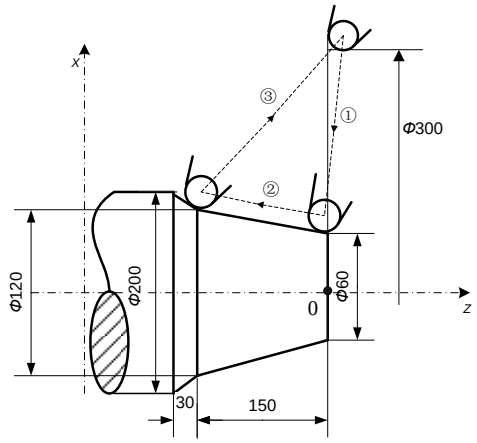


Figure II-52

(In G40 mode, radius programming)

G42 G00 X3.0;

G01 X6.0 W-15.0 F100 ;

G40 G00 X15.0 W15.0 ;

7.3.4 Precautious about Tool Nose Radius Compensation

Don't program two or more blocks continuously without a movement command.

Blocks without a movement command are:

- M05 ;—M code output
- S21 ;—S code output
- G04 X1000 ;—Dwell.
- G01 U0 ; —.Movement distance zero
- G98 ;.—G code only
- G10 P01 X100 Z200 R50 T2 ;—Offset change

If two or more of the above blocks are specified continuously, the tool nose center comes to a position vertical to the previous block programmed path at the end of the previous block. However, if the movement command is above 4), the above tool movement is attained only using one block.

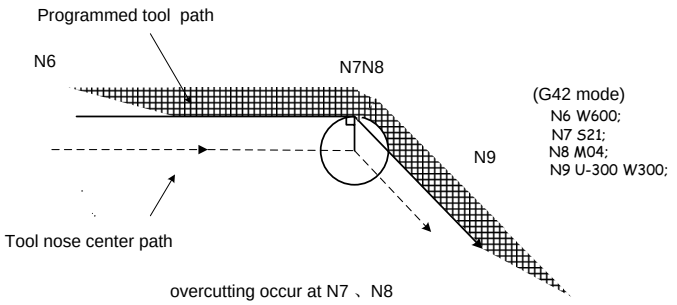


Figure II-53

Compensation with G90 or G94

Tool nose compensation with G90 or G94 is as follows:

1 As for every cycle path, tool nose center path is generally parallel to the programmed path.

(1) G90

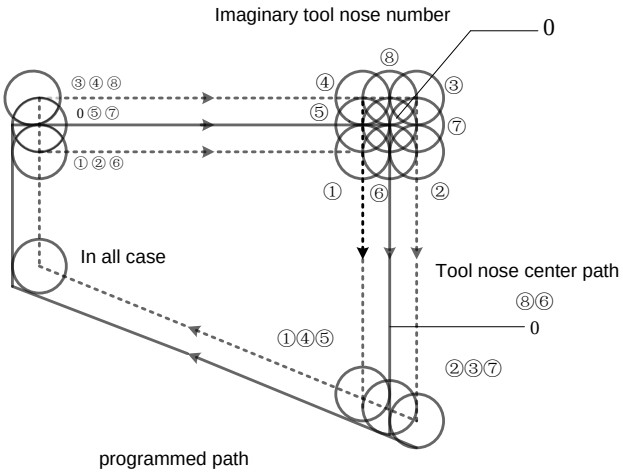


Figure II-54

(2) G94

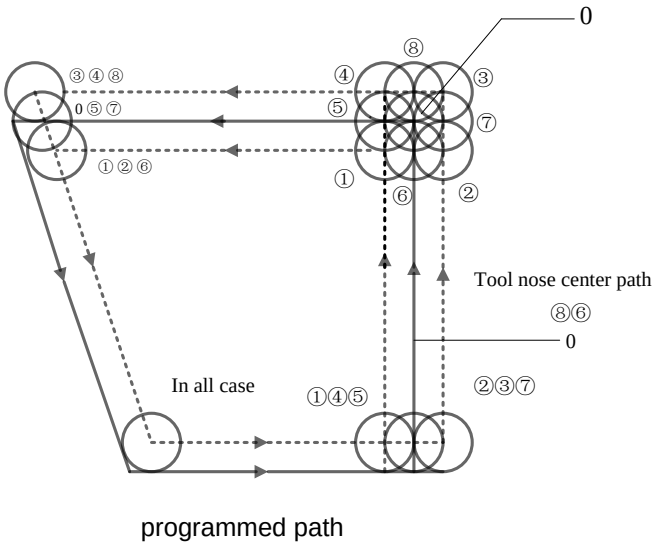


Figure II-55

2 Regardless of the G41 and G42 mode, offset direction is as shown in below figures.

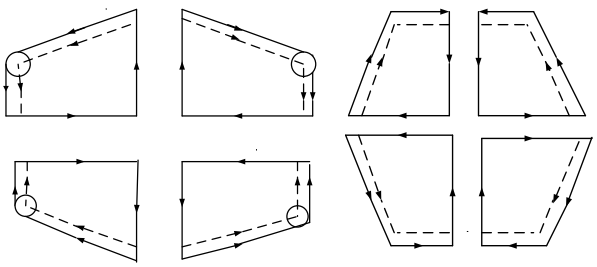


Figure II-56

3 Compensation with G71, G72 or G73

When one of above cycles is specified, the path deviates by a tool nose radius vector. During the cycle, no intersection calculation is performed.

4 G74~G76

Tool nose radius compensation is not performed in this case.

5 At chamfering

The movement after compensation is as shown in Figure II-57.

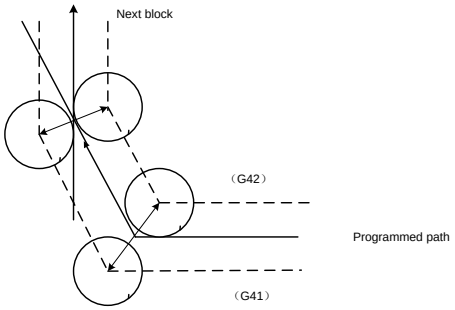


Figure II-57

6 When a corner circular is inserted

Movement after compensation is as shown in Figure II-58

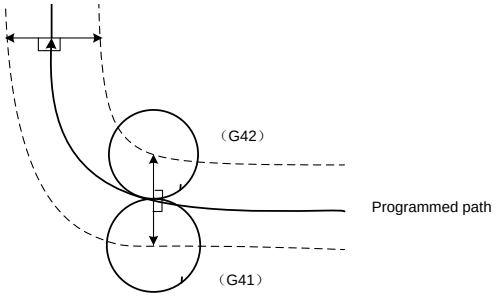


Figure II-58

7 When the block is specified in the MDI mode

In this case, tool nose radius compensation is not performed.

1 Inside corner is smaller than the tool nose radius (as shown in Figure II-59)

In this case, the inner offset of the tool will result in overcutting.

The tool will stop and ALARM (№41) is displayed just after starting the previous block, or after the corner movement if any. However, if the SINGLE BLOCK is ON, the tool stops at the end of the previous block.

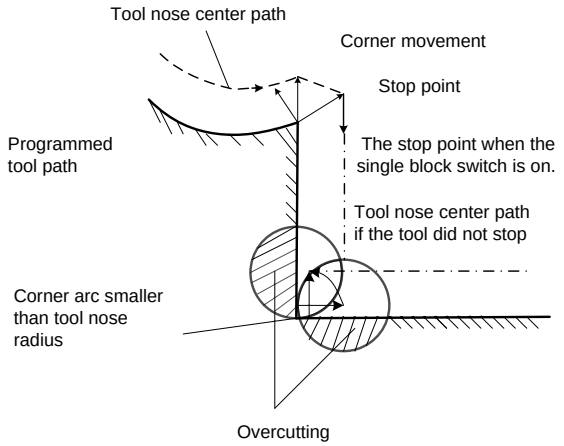


Figure II-59

8 Machining a slot smaller than the tool nose diameter (as shown in Figure II-60)

When the tool nose radius compensation makes the path of the tool nose center move in a direction opposite to the programmed path, overcutting will occur. In this case, ALARM (№41) is displayed and the motion stops just after starting the previous block (or after the corner movement), and the motion stops.

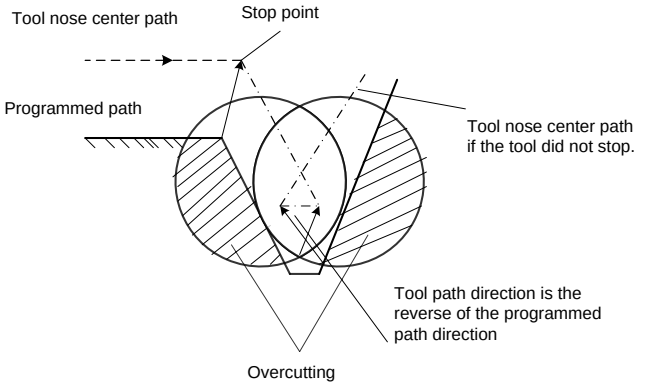


Figure II-60

9 When machining a step smaller than the tool nose radius (as shown in Figure II-61)

When a program contains a step smaller than the tool radius and this step is an circular, the path of the tool center may stroke reverse to the programmed direction. In this case, the first vector is ignored, and the tool moves linearly to the second vector position. The tool may stop at this point by the Single Block mode. If not machining in Single Block mode, cycle operation will continue.

If the step is a line, the offset is properly performed without generating any alarm. (However, the uncut part will still remain.)

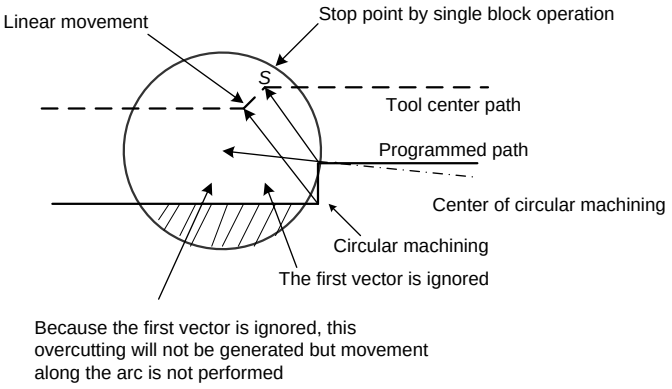


Figure II-61

7.3.5 Details about Tool Nose Radius Compensation

Tool nose R center offset vector

The tool nose R center offset vector is a 2D vector whose value is equal to the offset value specified by T code, and is calculated in the CNC. Its direction changes according to the tool movement block by block. This offset vector (hereinafter short as "vector") is internally created by the control unit as required for proper offsetting and to calculate a tool path with exact offset (by tool nose radius) from the programmed path. This vector is deleted at reset.

The vector always accompanies the tool motion. Proper understanding of the vector is essential for correct programming. Read carefully the description below about how vectors are created.

G40, G41, G42

G40, G41, G42 is used to delete or generate vectors, these codes and G00, G01, G02, G03 are used to set a mode for tool movement offset.

- **Cancel mode**

After power ON, as soon as 

is pressed or at a program end by executing M30, the system enters into the Cancel mode immediately.

In the Cancel mode, the vector is set to zero, and the path of the center of tool nose is identical with the programmed path. A program must end in Cancel mode. If program does not end in the Cancel mode, the tool can't be positioned at the end point, and the tool stops at a location that is a vector length away from the end point.

- **Offset start-up**

When a block satisfying all the following conditions is performed in Cancel mode, the system enters the Offset mode. Control during this operation is called Offset Start-up.

- The block contains G41 or G42, or has been specified to G41 or G42 mode.
- The offset number for tool nose radius compensation is not 00.
- X or Z movement is specified in the block and the move distance is not zero.

A circular command (G02 or G03) shall not be allowed in a block where the compensation starts, otherwise, ALARM (№34) will occur.

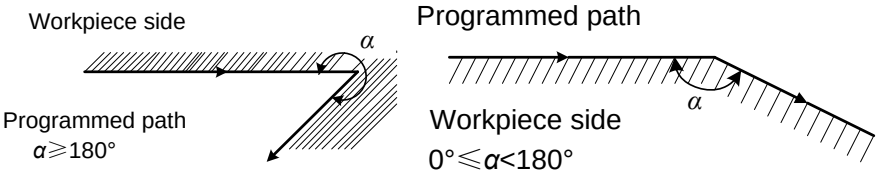
Two blocks are read during start-up. The first block is performed, and the second block is entered into the tool nose radius compensation buffer.

In the Single Block mode, two blocks are read, the first block is performed, and then the machine stops.

In continuous operations, two blocks are read in advance, so the CNC has one block currently being performed and the next two blocks.

NOTE

The meanings of "inner-side" and "outer-side" that occur frequently hereinafter are as follows: when the angle of intersection created by two blocks of move commands measured at the workpiece side is over 180°, it is "inner-side", as shown in Figure II-62; and when the angle is between 0 °and 180 °, it is an "outer-side" as shown in Figure II-62

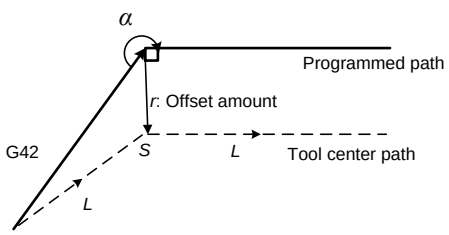


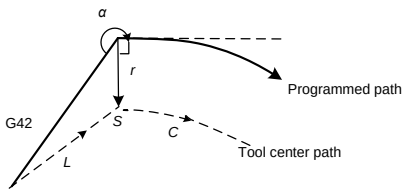
a)Inner side b)Outer side

Figure II-62

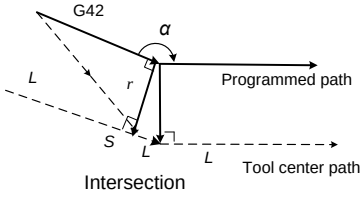
In below figure, S: intersecting point; L: linear; C: circular

a. When the tool goes around inside of corner(α≥180°)

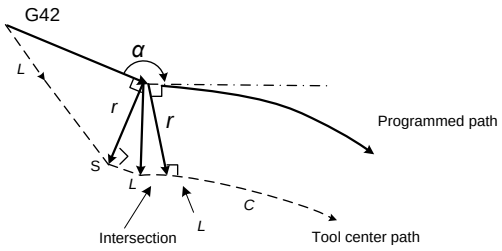




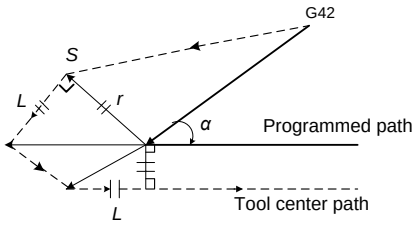
b. When the tool goes around the outside of corner (180 °>α≥90°)

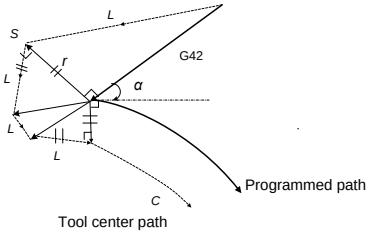


The intersection is the point where two offset paths calculated by continuous two blocks are Intersected

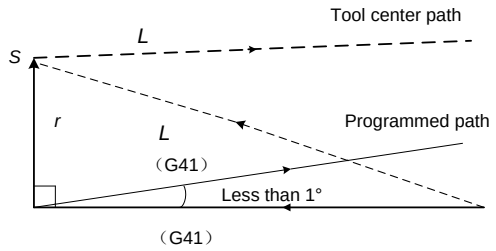


c. When the tool goes around the outside of corner (α<90 °)





d. When the tool goes around the outside of corner (α＜1°)

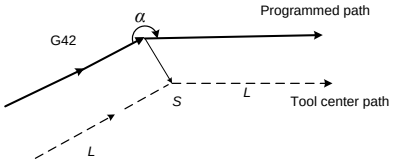


Offset mode

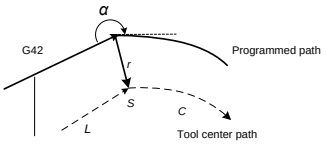
In the Offset mode, if non-positioning commands (such as auxiliary functions or dwell) have not been programmed in two or more successive blocks, offset can execute correctly; otherwise overcutting or undercutting will occur.

a. When the tool goes around inside of corner (α≥180°)

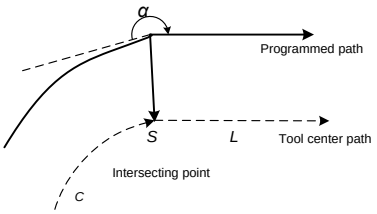
Ⅰ）Linear→Linear



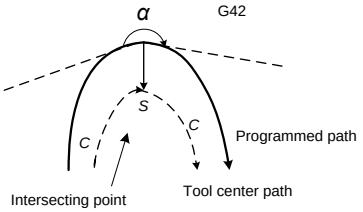
Ⅱ）Linear→Arc



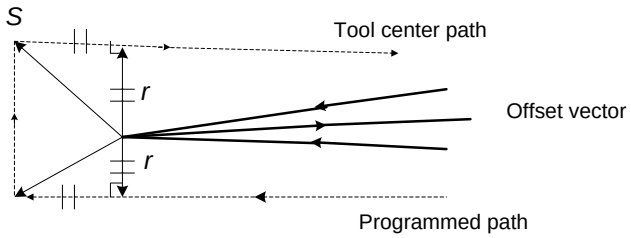
Ⅲ）Arc→Linear



Ⅳ）Arc→Arc



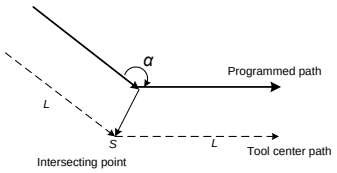
Ⅴ）Straight line→Straight line in the case in which it goes around at a narrow angle (less than 1°) and the offset vector became abnormally large.



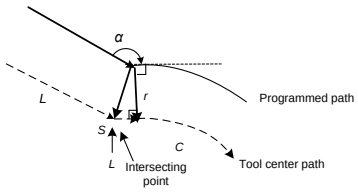
Also in case of arc to straight line, straight line to arc and arc ,the reader shoulder Infer infer in the same procedure

b. When the tool goes around outside of corner (180°＞α≥90°)

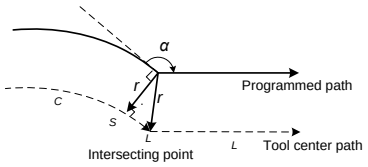
Ⅰ) Linear→Linear



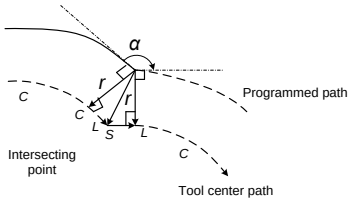
Ⅱ) Linear→Arc



Ⅲ) Arc→Linear

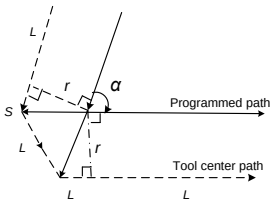


Ⅳ) Arc→Arc

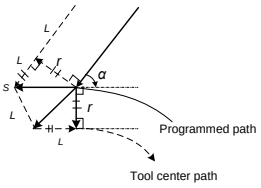


c. When the tool goes around outside of corner (α＜90°)

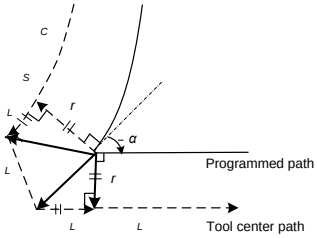
Ⅰ) Linear→Linear



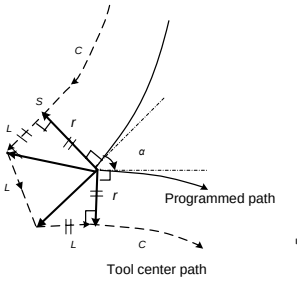
Ⅱ) Linear→Arc



Ⅲ) Arc→Linear

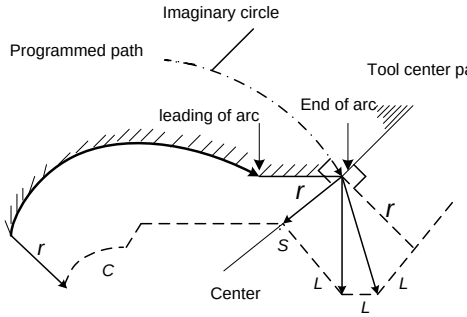


Ⅳ) Arc→Arc



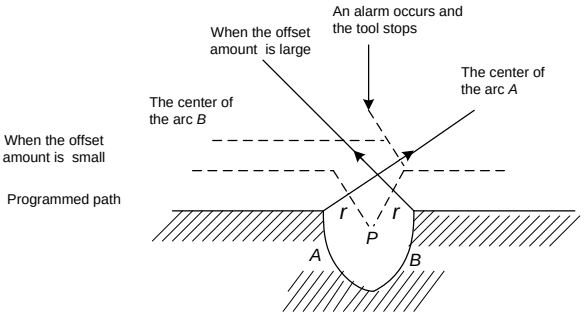
d. Exceptional cases

Ⅰ) The end point of arc is not on the arc



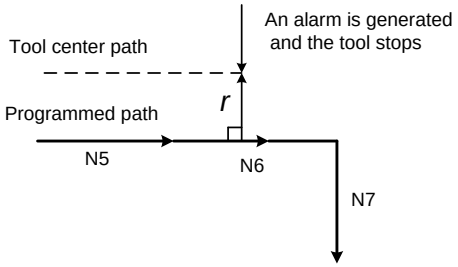
If there is an arc, lead-in line as shown, assume an imaginary circle which passes through the arc end point and is concentrically to the arc, and draw vectors for compensation, on the assumption that the cutter compensation has been carried out for the imaginary circle, As a result, a path for the cutter center, is established, which is different from the path compensated according to the program that regards the arc lead-in line as a straight line.
Similarly consider a case of arc-arc.

Ⅱ）There is no inner intersecting point



In regard to the left figure, the intersecting point of the arcs is present on the offset path if the offset value is small，but it may disappear if the offset value is quite large.In this case, alarm(No.33) occurs at the end point of the preceding block,and the tool will stop.
Concerning the left figure, there is an intersecing point *P* of the offset paths for the arcs *A* and *B*,if the offset value is small ,but the intersecting point can not be found if the offset value is quite large.

Ⅲ）The center of the arc is identical with the start point or end point



If the center of the arc is identical with the start point or end point，Alarm No.38 is displayed ， and the tool will stop at the end point
of the preceding block.

(G41 mode)
N5 G01 W1000;
N6 G02 W1000 I0 K0;
N7 G03 U-1000 I-1000;

Offset Cancel

In the Offset mode, when a block satisfying any of the following conditions is performed, the system enters into the Offset Cancel mode, and the action of this block is called the Offset Cancel.

1 G40 has been commanded.

2 Tool radius compensation number is specified as 0

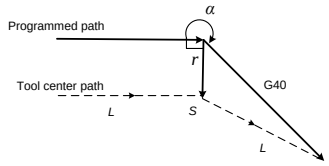
When executing Offset Cancel, circular commands (G02 and G03) are not available. If a circular is commanded, an ALARM (№34) will occur and the tool will stop.

In the Offset Cancel mode, the controller executes the commands in that block and the block in the tool radius compensation buffer. In the meantime, in the case of a Single Block mode, after reading one block, the controller executes it and stops. By pressing  once, one block is performed without reading the next block.

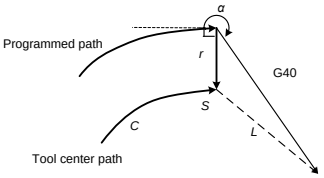
Then the controller is in the cancel mode, and normally, the block to be performed next will be stored in the buffer register and the next block is not read into the buffer for tool radius compensation.

a. When tool goes around inside of corner ($\alpha \geq 80°$)

Ⅰ) linear→linear

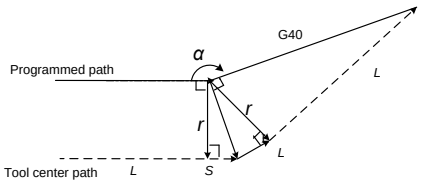


Ⅱ) Arc→linear

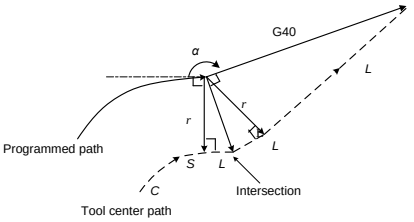


b. When tool goes around outside of corner (180°>α≥90°)

Ⅰ) linear→linear

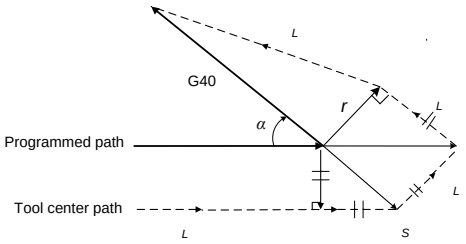


Ⅱ) Arc→linear

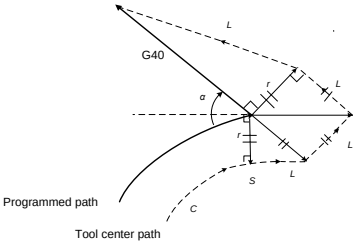


c. When tool goes around outside of corner(α<90°)

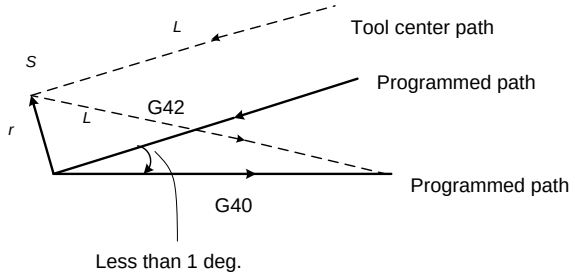
Ⅰ) linear→linear



Ⅱ) Arc→linear



d. When the tool goes around the outside of corner (α<1°)



Change offset direction in Offset mode

The offset direction is decided by G codes (G41 and G42), and the signs of offset are as shown in Table II-2.

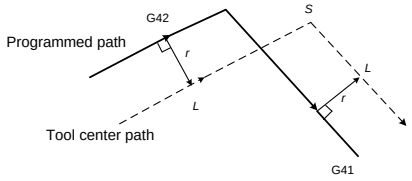
Table II-2

Sign of offset / G code	+	-
G41	Left side compensation	Right side compensation
G42	Right side compensation	Left side compensation

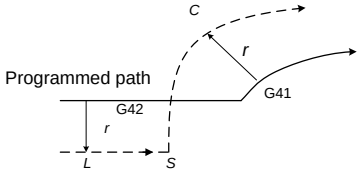
In a special case, the offset direction may be changed in the offset mode. However, the change is not available in the startup block and the block following it. When the offset direction changes, there is no concept of inside and outside. The offset amount in the following example is assumed to be positive.

Change offset direction in Offset mode

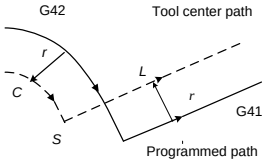
Ⅰ） linear→linear



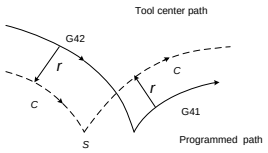
Ⅱ） linear→Arc



Ⅲ） Linear→Arc



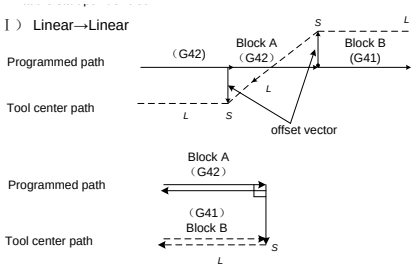
Ⅳ） Arc→Arc



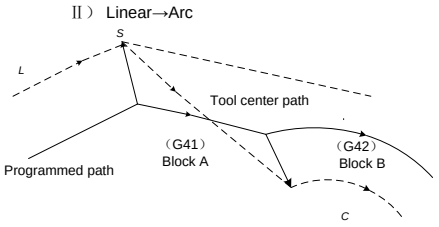
When an intersection is not obtained if offset is normally performed.

When G41 and G42 change the offset direction of block A ~ block B, intersection of the offset path is not required, the vector vertical to block B is created at the start point of block B.

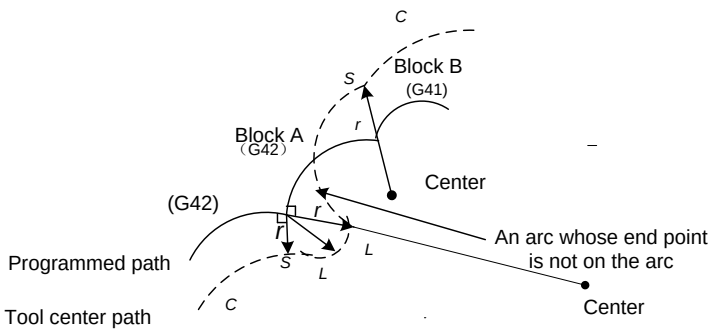
Ⅰ） Linear→Linear



Ⅱ） Linear→Arc



Ⅲ） Linear→Linear



Tool compensation G code in Offset mode

In Offset mode, by commanding the tool radius compensation G code (G41,G42), a vector in a right angle to the moving direction in the previous block will be formed, irrespective of machining on inner or outer side. If this G code is specified in a circular command, no correct circular will be obtained.

For changing the direction of offset by using the command of tool radius compensation G code (G41,G42), refer to 5.

Linear→Linear Circular→Linear

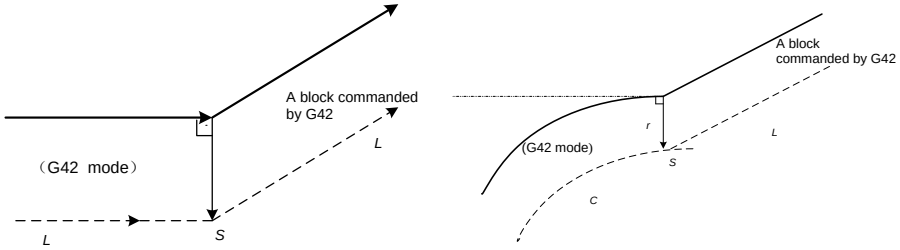


Figure II-63

Temporary offset cancel

If the commands below are specified in the Offset mode, the offset will be temporarily cancelled and thereafter the system will restore the offset mode automatically. Refer to Offset Cancel and Start-up for details of this operation.

G28 Automatic return to reference point

If G28 is commanded in the Offset mode, the offset will be cancelled at the intermediate point, and the Offset mode will be automatically restored after returning to the reference point.

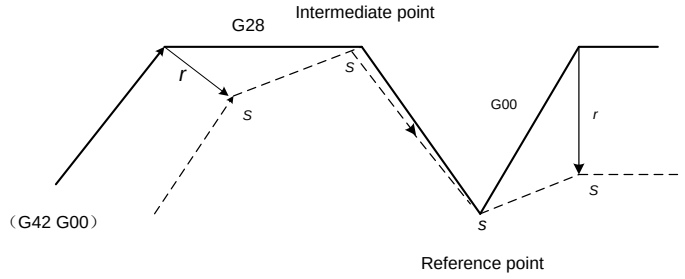


Figure II-64

Command for temporarily canceling offset vector

In Offset mode, if the following commands are specified, the offset vector will be temporarily cancelled and thereafter offset vector will be automatically restored. In this case, different from Offset Cancel mode, the tool moves directly from the intersecting point to the commanded point where the offset vector is canceled. When the Offset mode is restored, the tool moves directly to the intersecting point.

- **Coordinate system setting (G50)**

(G41 mode)

N5 G01 U3000 W7000;

N6 U－3000 W6000;

N7 G50 X1000 Z2000;

N8 G01 X4000 Z8000;

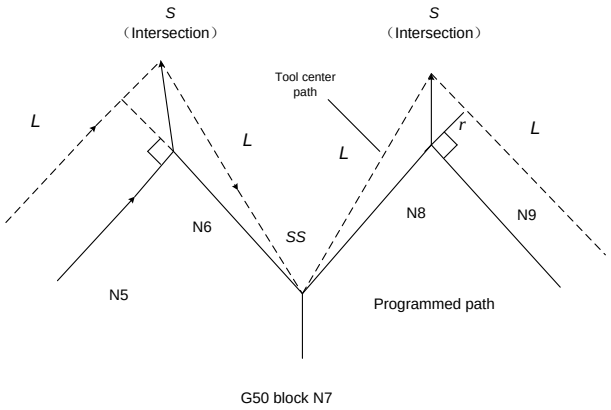


Figure II-65

NOTE

SS means the point where the tool stops twice in the Single Block mode.

- **G90, G92, G94 canned cycles, G71~G76 multiple repetitive cycles**

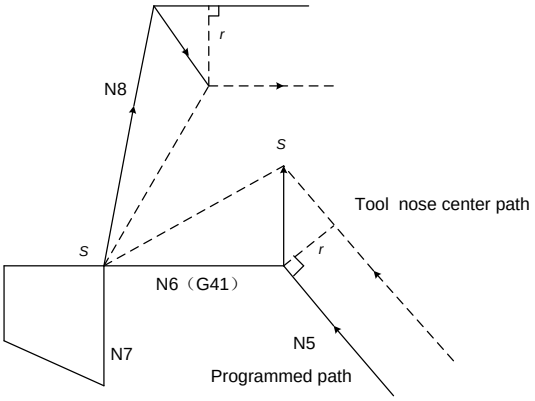


Figure II-66

(G42 mode)

N5 G01 U5000 W6000；

N6 W－8000；

N7 G90 U－6000 Z－8000 I－3000；

N8 G01 U12000 W5000；

- **Block containing T code**

A block without tool movement

The following blocks have no tool movement. In these blocks, the tool will not move even under tool radius compensation mode.

- M05 ; —— M code output.
- S21 ; —— S code output.
- G04 X10000 ; —— Dwell
- G01 P01 X100 ; —— Offset value change
- G98 ; —— G code only
- G01 U0;——Move distance is zero

- **When commanded at start-up**

If a block without tool movement is commanded at start-up, the offset vector is not produced.

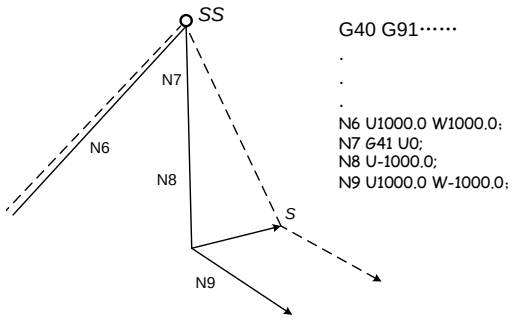


Figure II-67

- **When commanded in offset mode**

When a block without tool movement is commanded in the Offset mode, the vector and tool center path are the same as that when the block is not commanded. (Refer to item (3) Offset mode) This block is performed at the Single Block stop point.

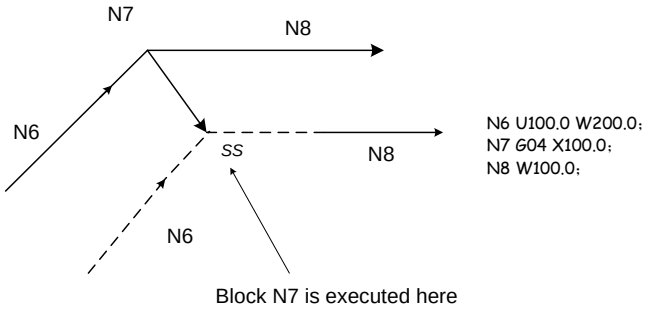


Figure II-68

However, when the move distance is zero, even if only one block is commanded, tool motion becomes the same as that when more than one block without tool movement are commanded, which will be described in detail later.

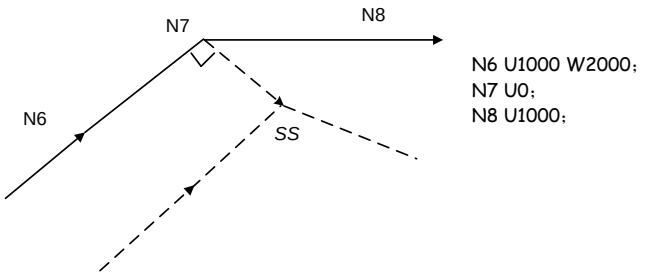


Figure II-69

Two blocks without tool movement should not be commanded consecutively. If commanded, a vector whose length is equal to the offset value is produced in a direction vertical to the tool motion in the previous block, so overcutting may result.

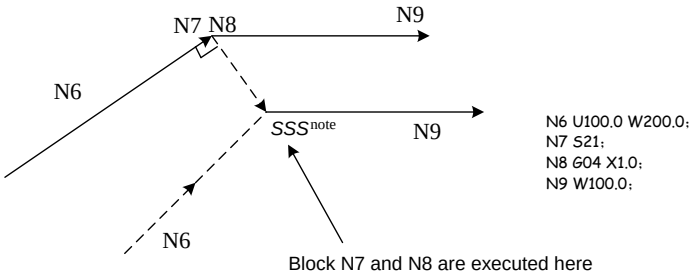


Figure II-70

NOTE

SSS means that tool stops three times by Single Block operation.

- **When commanded together with Offset Cancel**

When a block without tool movement is commanded together with an Offset Cancel command, a vector whose length is equal to the offset value is produced in a direction vertical to tool motion in the previous block, and the vector will be cancelled in the next move command.

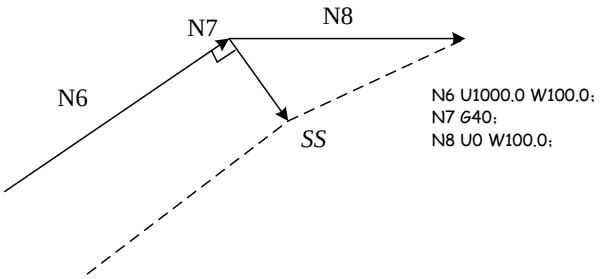


Figure II-71

Corner movement

When two or more vectors are produced at the end of a block, the tool will move linearly from one vector to another. This movement is called corner movement.

If these vectors almost coincide with each other, the corner movement will not be performed and the latter vector is ignored.

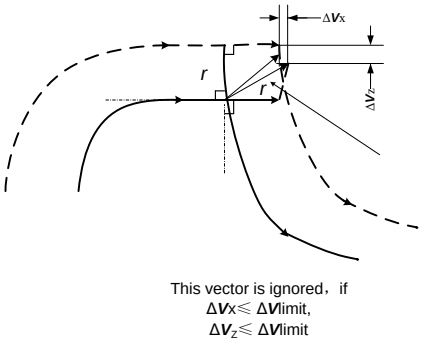


Figure II-72

If ΔVX is ≤ΔV limit and ΔVZ is ≤ΔV limit, the latter vector is ignored. The ΔV limit is set in advance by CRCDL of parameter N0.049.

If these vectors do not coincide, a move is generated around the corner. This move belongs to the latter block.

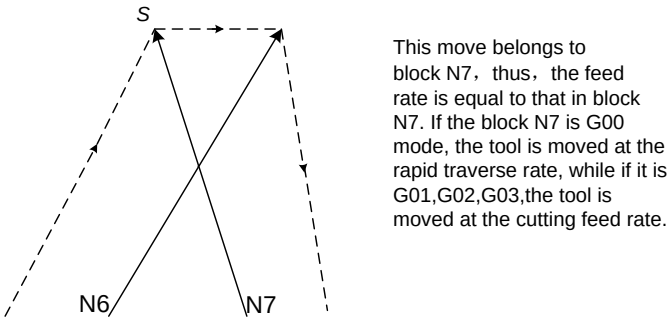


Figure II-73

Interference check

Tool overcutting is called “interference”. The interference check function checks for tool overcutting in advance. However, this function cannot check all interferences. The interference check is performed even if overcutting does not occur.

- **Basic conditions for interference**

1 The direction of the tool path is different from that of programmed path (the included angle is 90 °~270 ° between these paths).

2 For circular machining, in addition to the above condition, the angle between the start point and end point on the tool center path is quite different from that between the start point and end point on the programmed path (More than 180 °).

Example 1

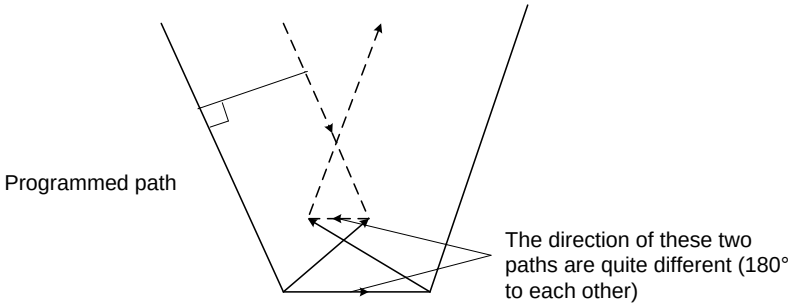


Figure II-74

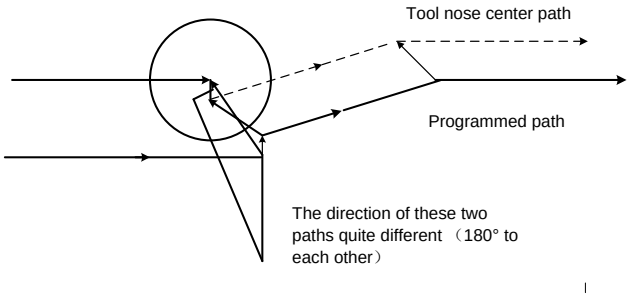


Figure II-75

Example 2

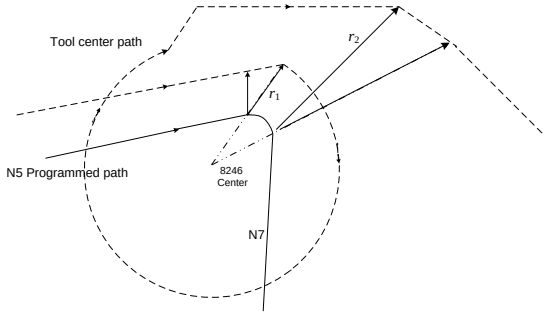


Figure II-76

(G41)

N5 G01 U2000 W8000 T1;

N6 G02 U－1600 W3200 I－1800 K -28000 T2;

N7 G01 U－1500 W-2000 ;

(Offset value corresponding to T1: r1 =2000)

(Offset value corresponding to T2: r2 =6000)

In the above example, the circular of block N6 is placed in one quadrant. But after tool compensation, the circular is placed in four quadrants.

- **Correction of interference in advance**

1 Ignoring the vector causing the interference

When tool compensation is performed for blocks A, B and C, vectors V1,V2,V3 and V4 between blocks A and B, and V5, V6, V7 and V8 between B and C are produced, the nearest vectors are checked first. If interference occurs, they are ignored. But if the vectors to be ignored are at the last of the corner, they cannot be ignored.

Interference check:

Between V4 and V5 —— interference —— V4, V5 Ignored

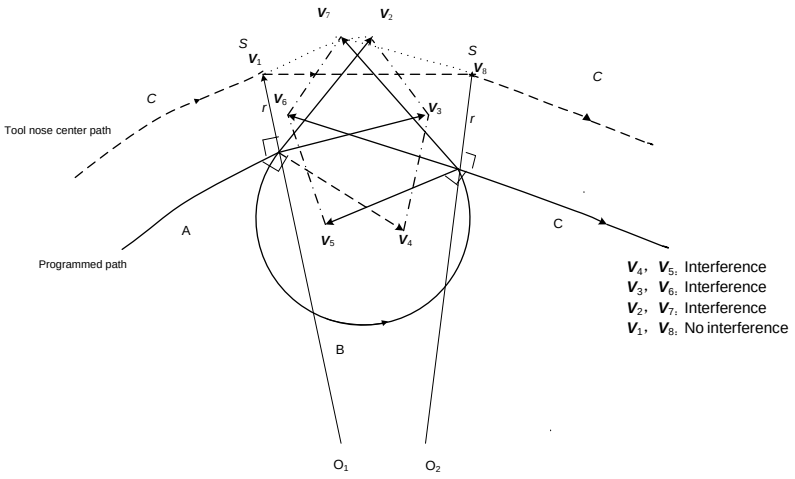
Between V3 and V6 —— interference —— V3, V6 Ignored

Between V2 and V7 —— interference —— V2, V7 Ignored

Between V1 and V8 —— interference —— V1, V8 cannot be ignored

If while checking, a vector with no interference is detected, subsequent vectors are not checked. If block B is in circular movement, a linear movement is produced if the vectors are interfered.

Example 1 The tool moves linearly from V1 to V8 (as shown in Figure II-77)

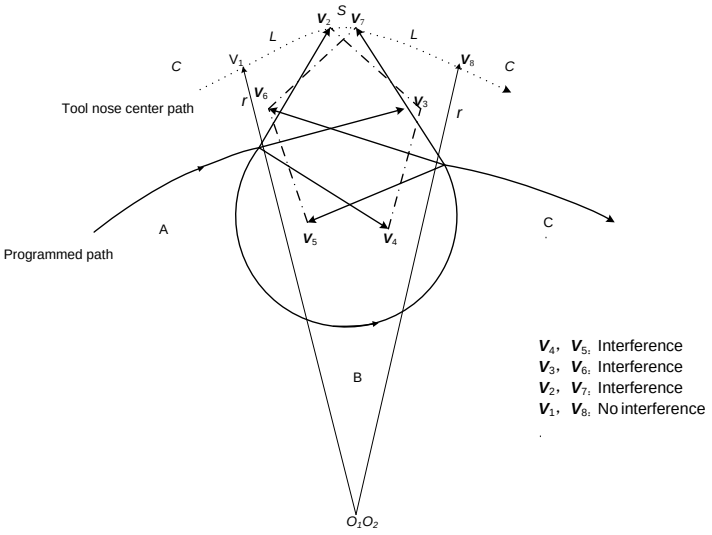


If the tool is stopped by Single Block A, the tool center moves to V_3.Then starting operation will move the toool to V_7 or V_8.

Figure II-77

Example 2: The tool moves linearly as follows (as shown in Figure II-78):

Tool path : V1 → V2 → V7→ V8



If the tool is stopped by single block operation at block A, the tool center moves to V_3.Then putting the operation into start moves the tool to V_7 or V_8.

Figure II-78

2 If the interference still occurs after above correction step, the tool is stopped with an alarm. If the interference occurs after above correction or if there are only one pair of vectors from the beginning of check and the vectors interfere, the ALARM (№4) is displayed and the tool is stopped immediately after execution of the previous block. (If the block is performed by the Single Block mode, the tool is stopped at the end of the block.)

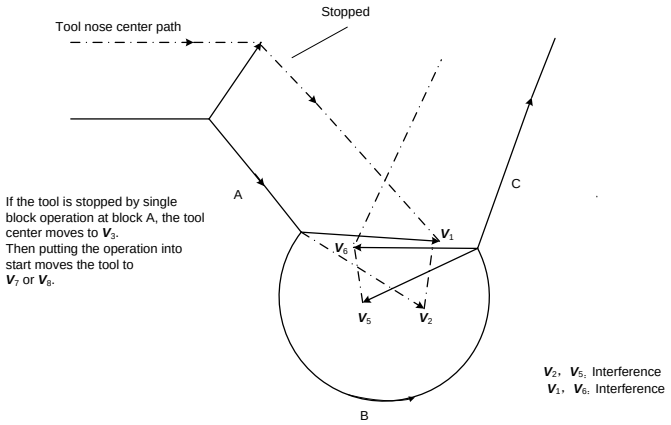


Figure II-79

After ignoring vectors V_2 and V_5 because of interference, interference also occurs between vectors V_1 and V_6. The alarm is displayed and the tool is stopped.

3 When interference is supposed though there is no actual interference

Here several examples are given:

Example 1: Depth is smaller than the offset value (as shown in Figure II-80)

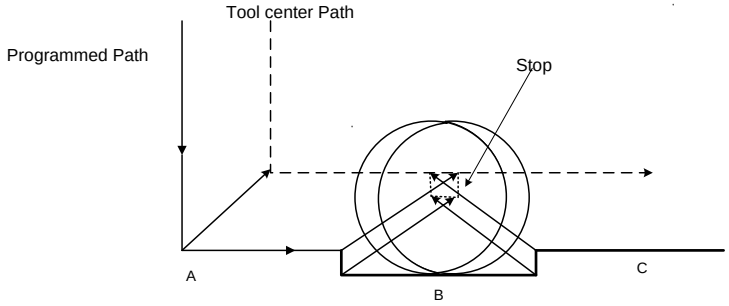


Figure II-80

There is no actual interference, but since the direction programmed in block B is opposite to that of the path after tool compensation, the tool stops and an alarm is displayed.

Example 2: Slot is smaller than the offset value (Figure II-81)

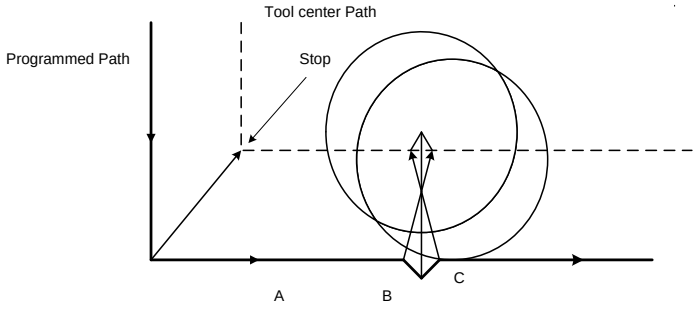


Figure II-81

Like Example 1, the direction is reverse in block B.

Correction in chamfering and corner circulars

1 In chamfering or for corner circulars, tool nose radius compensation will only be performed when an ordinary intersection exists at the corner. In Offset Cancel mode, no compensation can be performed at

offset start-up or when the offset direction changes, in this case, an ALARM (№039) will occur and the tool will stop.

2 In inner chamfering or for inner corner circulars, if the chamfering value or corner circular value is smaller than the tool nose radius value, the tool is stopped with an ALARM (№039) since overcutting will occur.

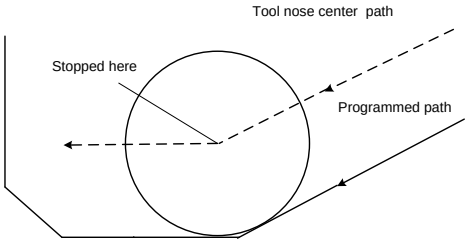


Figure II-82

3 Undercutting or alarming

Example 1:The following example shows undercutting.

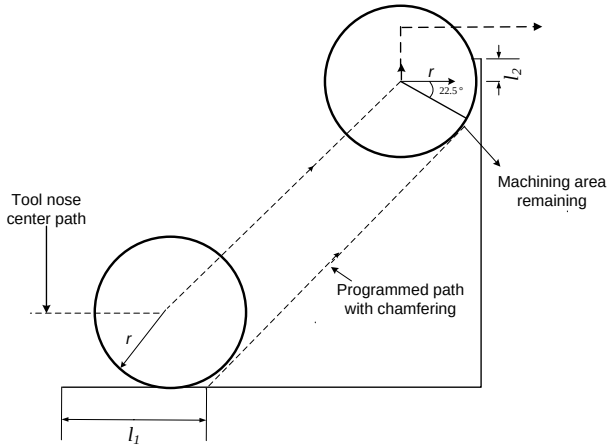


Figure II-83

In inner chamfering, if the length of the programmed path that is not a part of the chamfering (in the Figure II-83 l_1 and l_2) is in the following range, undercutting will occur.

$0 \leq l_1, l_2 \leq r \times \tan 22.5°$

(r: tool nose radius)

Enlarged view on the undercut area is as shown in Figure II-84:

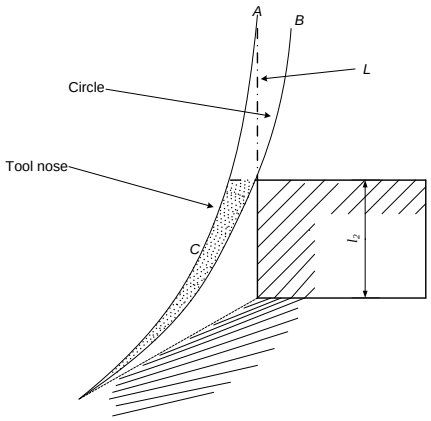


Figure II-84

Although the tool should be positioned at A in the above figure, the tool is positioned at B (the tool nose is tangent to line L), thus, area C is not machined.

Example 2: ALARM (№52) or ALARM №55 is displayed in the following cases (as shown in Figure II-85):

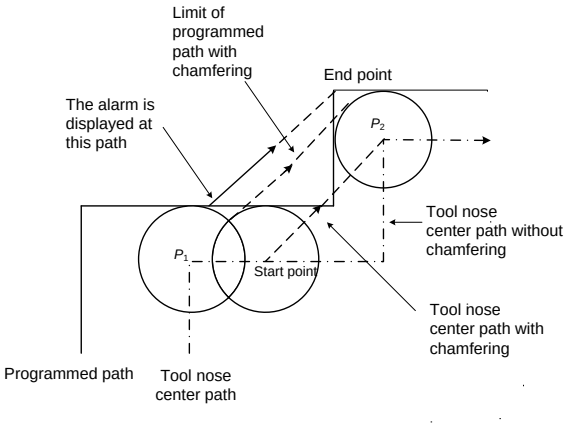


Figure II-85

In outer chamfering with an offset, a limit is imposed on the programmed path. The path during chamfering coincides with the intersection points P1 or P2 without chamfering, therefore, outer chamfering is limited. If the chamfering value is more than the limit value specified, ALARM (№52) or ALARM(№055)will be displayed.

Input command in MDI

Compensation is not performed for commands input in the MDI.

However, the tool path is as follows after inserting MDI operation and starting automatic operation again if the automatic operation using NC program written by absolute commands is temporarily stopped by the Single Block function:

In this case, the vectors at the start point of the next block are transmitted and other vectors are produced by the next two blocks. Therefore, from point after PC, compensation can be accurately performed.

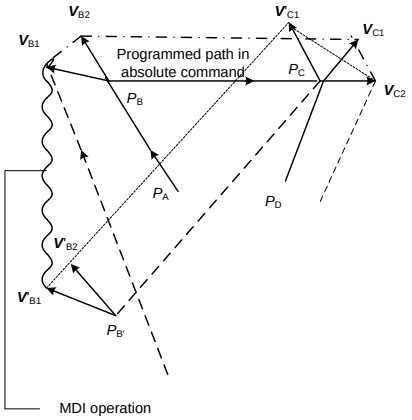


Figure II-86

When points P_A, P_B and P_C are programmed in an absolute command, the blocks is stopped by the Single Block function after executing the block from P_A to P_B and the tool is moved by MDI operation. Vectors V_{B1} and V_{B2} are transmitted to V'_{B1} and V'_{B2} and offset vectors are recalculated for the vectors V'_{C1} and V_{C2} between block $P_{B'} \rightarrow P_C$ and $P_C \rightarrow P_D$.

However, since vector VB2 is not calculated again, compensation is accurately performed from point PC.

Manual operation

For manual operation during the tool compensation, refer to section 3.3.

Subprogram

Before calling a subprogram, namely, before M98 command is performed, the system must be in Offset Cancel mode. Offset can be started after going into the subprogram, but before returning to the main program, i.e. executing M99 command, it must be in Offset Cancel mode; otherwise ALARM (№.36) will occur.

General precautious for offset operations

1 Changing the offset value

In general, the offset value is changed in Cancel mode or when changing tools. If the offset value is changed in Offset mode, the vector at the end point of the block is calculated for the new offset value.

2 The polarity of the offset amount and the tool nose center path

When a negative offset value is specified, the program is performed such that G41 and G42 are exchanged. A tool machining an inner profile will machine the outer profile, and tool machining the outer profile will machine the inner profile.

An example is shown below. In general, CNC machining is programmed assuming a positive offset value. When a program specifies a tool path as shown in 1, the tool will move as shown in 2 if a negative offset is specified. The tool in 2 will move as shown in 1 when the sign of the offset value is reserved.

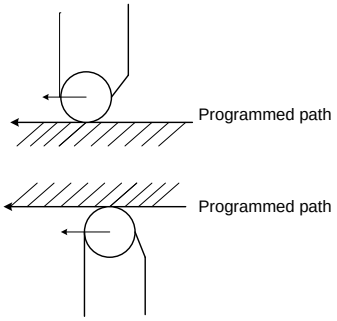


Figure II-87

NOTE

1 When the sign of the offset value is reversed, the offset vector of the tool nose is reversed but the imaginary tool nose direction does not change.

2 Therefore, do not reverse the sign of the offset value when starting the machining meeting the imaginary tool nose to the start point.

8 EXAMPLE

Machining a shaft (as shown in Figure II-88), the tool used is:

T01: outer diameter turning tool; T02: slotting tool, tool width: 3mm; T03: 60 °thread turning tool.

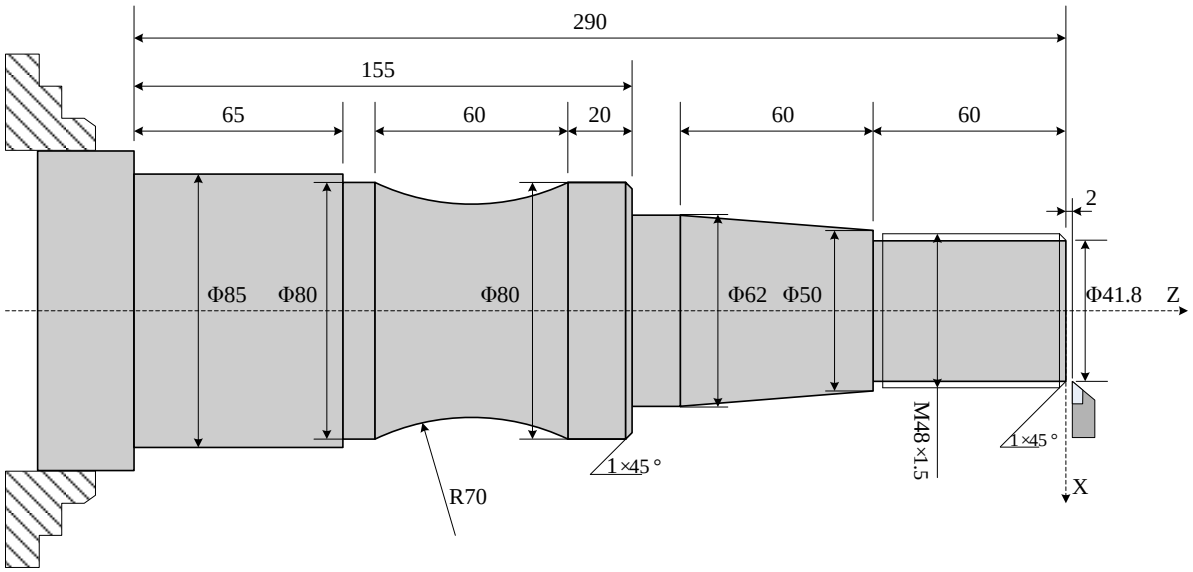


Figure II-88 Example of synthetic programming

The program is as follows:

N10 M03 S××;	Spindle on
N20 T0101;	1st tool is selected, and compensation on the 1st tool is carried out
N30 G00 X41.8 Z2 M08;	Traversing to the point ready for machining; cutting fluid on
N40 G01 X48 Z-1 F100;	Chamfering
N50 Z-60;	Finish turning the bigger thread diameter
N60 X50;	Retracting
N70 X62 W－60;	Finish turning the taper surface
N80 W－15;	Finish turning Φ62mm outer diameter
N90 X78;	Retracting
N100 X80 W-1;	Chamfering
N110 W-19;	Finish turning Φ80mm outer diameter
N120 G02 X80 W-60 R70;	Finish turning circular (I, K is I63.25 K-30)
N130 G01 Z-225;	Finish turning Φ80mm outer diameter
N140 X85;	Retracting
N150 Z-290;	Finish turning Φ85mm outer diameter
N160 X90 M09;	Retracting, cutting fluid off
N170 G00 X150 Z50;	Traversing to the tool-change point
N180 T0202;	Tool changing, setting up 2# tool compensation
N190 M03 S××;	Spindle speed varying
N200 G00 X51 Z-60 M08;	Traversing to the machining point, setting the tool with the left tip of the tool
N210 G01 X45 F90;	Turning Φ45mm slot

N220 G00 X51; Retracting

N230 X150 Z50 M09; Returning to the tool-change point, cutting fluid off

N240 T0303; Tool changing, setting up tool compensation

N250 M03 S××; Spindle speed varying

N260 G00 X62 Z6 M08; Traversing to the point ready for machining; cutting fluid on

N270 G92 X47.54 Z－58 F1.5; Cycle for thread cutting

N280 X46.94;

N290 X46.54;

N300 X46.38;

N310 G00 X150 Z50 M09; Returning to the tool-start point, cutting fluid off

N320 T0300; Cancel tool compensation

N330 M05; Spindle stop

N350 M30; Program end

III OPERATION

1 GENERAL

Before operating K100Ti-B, the following should be well understood .

Manual Operation

1 Manual return to reference or manual program homing.

2 Tool movement by manual mode

3 Manual miscellaneous functions

Automatic Operation

1 Memory operation: After a program is saved in the memory of CNC, the machine can be run according to the program command. This operation is called memory operation

2 MDI operation: After a program is entered as a command group from the MDI keyboard, the machine can be run according to the program. This operation is called MDI operation.

Edit

1 Store a part program into the memory of the CNC.

2 Edit and alter the program via  key on the panel in EDIT mode.

Test

Before starting machining, the self-check on the machine can be performed. It checks whether the program can operate the machine as desired. This check can be accomplished by running the machine actually or just viewing change in the displayed positions without running the machine.

1 Check by running the machine

- Feedrate override

  Check the program by changing the feedrate override.

- Single block

  When the  is pressed, the machine will execute one block and stop after the block is finished. The machine will execute next operation and stop by pressing  again. The program can be checked by this way.

2 With machine locked, view change in displayed positions or tool path via Graphic function.

Displaying and Setting Data

1 Display and set the offset value.

2 Display and set the parameters.

3 Judge the status of DI/DO signals by diagnose parameter

Display

1 Program display

2 Current position display

3 Alarm display

Operation of the Flash Memory

Graphic Function

2 OPERATION AND DISPLAY

2.1 Panel

K100Ti-B LCD/MDI panel is as shown in Figure III-1.

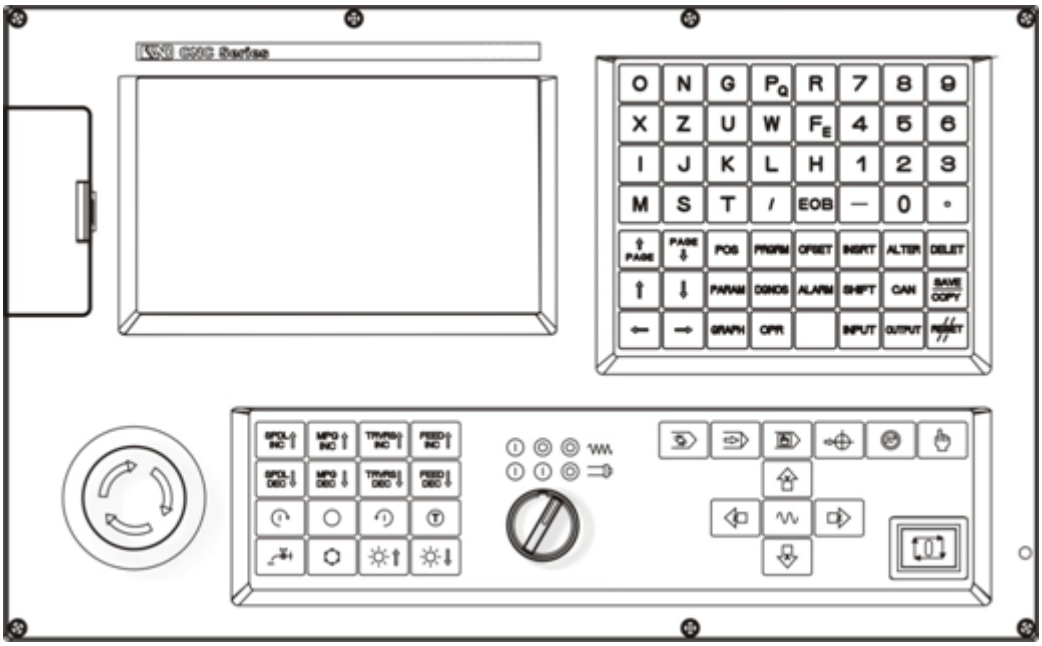


Figure III-1

2.1.1 LCD display area

K100Ti-B CNC system adopts a 7 inch TFT color LCD.

2.1.2 Serial port and U-disk interface area

For the user's convenience, there is COM in front and behind of the system and U disk function.

2.1.3 Edit keyboard area

Table III-1

Key	Name	Function description
	Reset key	CNC reset, feed, output stop

Key	Name	Function description
	Number keys	Inputting numbers
	Address keys	Inputting address symbols Note: P key is of multiple functions, can be used also as Q. F key is of multiple functions, can be used also as E.
	Page-up/Page-down key	Page up and page down
	Cursor keys	To control cursor movement.
	EOB key	To input block end symbol, namely input ";". Pressing the key, the program block ends and changes to a new line
	Edit keys	To insert, modify and delete character and program
	Input key	To start communication inputting
	Output key	To start communication outputting
	Save and copy key	For flash memory saving, and program copying
	Shift key	For selecting various displays, example: P'021~P'040 parameters

2.1.4 Display interface menu area

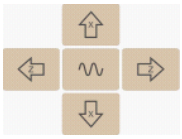
Table III-2

Menu key	Remarks
	Enter Position interface which includes four pages: relative coordinates, absolute coordinates, synthetical coordinates, coordinates & program
	Enter program interface which includes four pages: program content, program status, program directory, U-disk.
	Turning into Tool Compensation interface and Measure interface (repeatedly pressing the key can vary between two interfaces). The Tool Compensation interface can display tool offset values.
	Displaying Parameter page. By pressing repeatedly, it will display next page (identical to Page-down)
	Displaying Diagnostic page. Display is invariable even if pressing repeatedly.
	Displaying Alarm page. By pressing repeatedly, Alarm and EX-ALM page alternate.
	Displaying Graph page. By pressing repeatedly, Graph and Graphic Parameter page alternate.
	Displaying Test page. Display is invariable even if pressing repeatedly.

2.1.5 Machine tool panel area

Table III-3

Key	Name	Function description
	EDIT mode selection key	Turning into EDIT mode
	AUTO mode selection key	Turning into AUTO mode
	MDI mode selection key	Turning into MDI mode
	HOME mode selection	Turning into Mechanical Zeroing mode
	Step mode selection key	Turning into Step or Handwheel operation (one of the two modes is selected by parameter)
	JOG mode selection key	Turning into Manual mode
	Start key	Starting program and MDI command

Key	Name	Function description
	Manual feed key	JOG and Step mode, X, Z axis +/- movement
	Spindle override key	Spindle override 50％~120％ (interval of 10％)
	Handwheel increment key	Handwheel or Step increment selection
	Traverse Override key	Four steps of traverse override F0, 25％, 50％, 100％. Traverse Override key may select four overrides, percentage value can be displayed at down left corner of Position page. F0 is specified by PA26.
	Feed Override key	JOG mode: manual feedrate function enabled AUTO mode: feed override enabled
	Coolant On/Off	Under Manual /Handwheel/Step mode, pressing the key and Lock button, the switch will output ‘on→off→on...’
	Manual Tool Change	Under JOG /Handwheel/Step mode, pressing the key, the tool rest will turn and change the next tool.
	JOG on/off key	Under Manual /Handwheel/Step mode, holding down the key, the spindle will turn CW. As soon as the key is released, the spindle stops.
	Spindle Control key	JOG/Handwheel/Step mode, spindle turns CW/stops/turns CCW.
	Pause three-position switch (Feed Pause and Spindle Pause)	The knob has three positions: Left: normal (feed on and spindle on); Middle: feed pauses (feed pauses and spindle on); Right: spindle pauses, feed pauses (feed pauses and spindle pauses); When the knob is at left, pressing  , machining goes on
	Emergency Stop switch	Pressing Emergency Stop switch, the system resets, feed stops, and “NOT READY” alarm occurs. As soon as the switch is released, the alarm disappears, the system requires tool

Key	Name	Function description
		presetting again.

NOTE

1 After turning the spindle in Manual mode, pressing any spindle key (CW, CCW, Stop, JOG), the spindle will stop. When the spindle is running in AUTO mode, specifying reverse direction to current direction, alarm will occur, and the Dwell program will run.

2 Traverse override

Four steps of override in 100%, 50%, 25%, F0 can be used on the following speed:

- G00 traverse feed
- Traverse feed of canned cycle
- Traverse feed of G28
- Manual traverse feed
- Traverse of manual return to reference point

Example: If traverse feed rate is 6 m/min, override is 50%, the actual feed rate is 3 m/min.

2.2 Overview on the Operation Modes

K100Ti-B system has six operation modes: EDIT, AUTO, MDI, HOME, STEP/Handwheel and JOG.

1 EDIT mode: In EDIT mode, you can create, delete and modify the machining programs.

2 AUTO mode: In AUTO mode, program runs automatically.

3 MDI mode: In MDI mode, you can input parameters as well as input and execute blocks.

4 HOME mode: In Mechanical Home mode, you can home X and Z axis respectively.

5 STEP/Handwheel mode: In STEP (STEP/Handwheel) mode, CNC will move with the selected increment.

6 JOG mode: In JOG mode, you can carry out such operations as manual feeding, manual traversing, feed override adjusting, traverse override adjusting, spindle on/off, coolant on/off, lube on/off, spindle jogging and manual tool change, etc.

2.3 Adjusting Brightness of LCD

The brightness of the system LCD can be adjusted as desired by user. The method is by pressing either key of  at any mode.

NOTE

K100Ti-B CNC system is equipped with a TFT true color LCD display, which has temperature compensation function. Normally it is not necessary to adjust the brightness.

2.4 Modification and Setting of Display Interface and Data

K100Ti-B has 8 screens: e.g. Position and Program, etc., under each screen, many display pages are

provided. Each screen is independent from the operation modes.

2.4.1 Position screen

To enter into the Position screen, press the key [POS]. Under the screen, there are four pages, i.e. Absolute Coordinate, Relative Coordinate, Synthetic Coordinate, and Coordinate & Program, etc., you can look up by pressing [POS] in a repeated way, or look up by [↑ PAGE] or [PAGE ↓] button.

Display screen for absolute coordinates

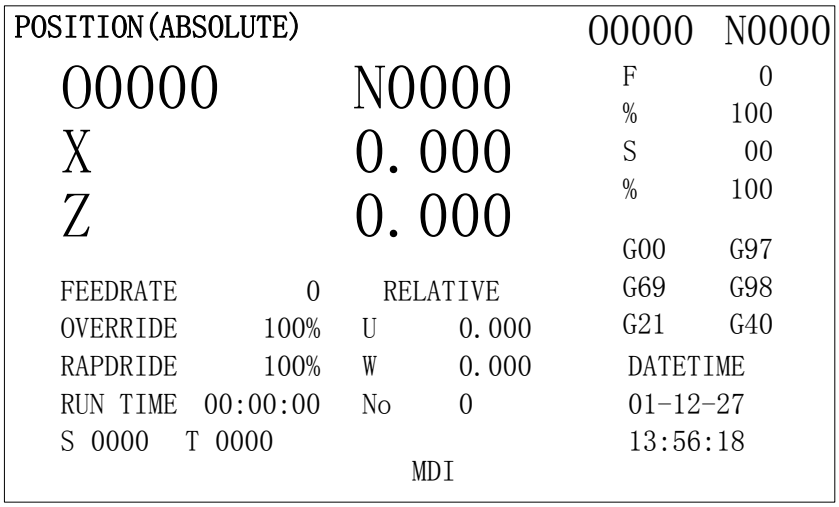


Figure III-2

The displayed X and Z coordinates are the absolute position of the tool in the current work coordinates system. When the CNC is turned on, the X and Z coordinates will remain as before, and the work coordinates system will be specified by G50.

1 FEEDRATE: the rate specified by F code in the program.
2 OVERRIDE: the override selected by the feed override switch.
3 RAPDRIDE: the G00 speed override selected by traverse override switch.
4 No (machined workpiece count): the machined workpiece counter increases 1 every time after the program executes M30.
5 RUN TIME: time is counted after the automatic running starts, the units of the time are hour, minute and second. The workpiece count and the cutting time will be memorized at power OFF.
6 S0000: The spindle speed feedback by the spindle encoder. The actual spindle speed can be shown only when a spindle encoder is mounted.
7 T0100: The current tool no. and the tool offset number.
8 Clock display: to display the current time.

NOTE

The “programmed rate” is the display in AUTO and MDI mode; “Manual rate” will be displayed in Mechanical Home and Manual mode; and the “Handwheel increment” is shown in Handwheel mode.

Display screen for relative coordinates

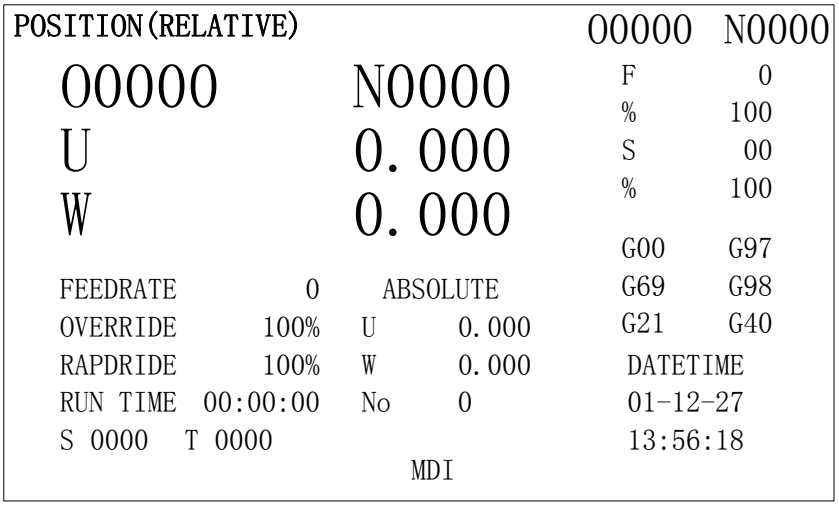


Figure III-3

The displayed U and W coordinates are the coordinates of current position relative to the reference point, which will remain when the CNC system power ON. U and W coordinates can be reset at any time. After resetting the U and W coordinates, the current point is a relative reference point.

Way for resetting U and W coordinates:

Press U key in the Relative Coordinate screen, U flickers in the screen, press CAN key, the U coordinate is reset; press W key in the Relative Coordinate screen, W flashes in the screen, press CAN key, the W coordinate is reset. After resetting, U and W will stop flickering.

Display screen for synthetic coordinates

In the Synthetic Position screen, it will concurrently display the relative coordinates, absolute coordinates, machine coordinates and the residual movement amount (only for Auto and MDI mode).

The displayed machine coordinates are the coordinates of the current position in the machine coordinates system, which is established by returning to the mechanical zero.

The residual movement amount is the differentiation between the target position of the block or MDI command and the current position.

The displayed page is as shown in Figure III-4:

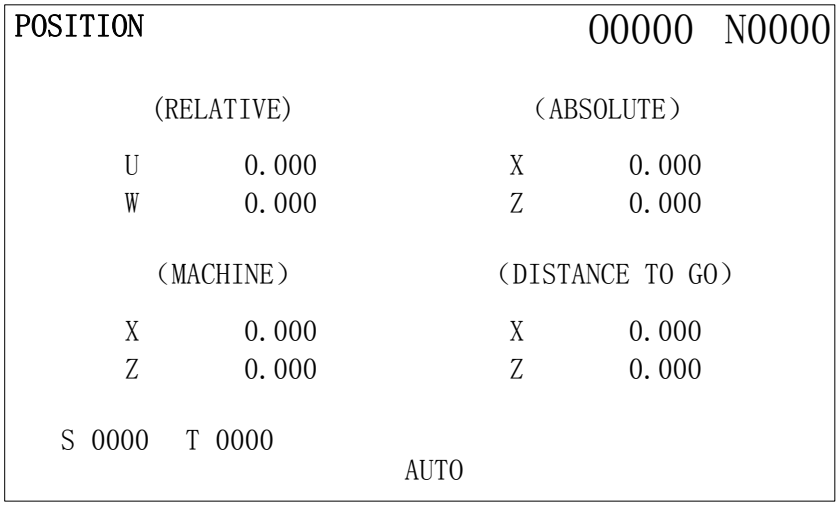


Figure III-4

Coordinate & Program display screen

In the Coordinate & Program display screen, it concurrently displays the absolute coordinates, relative coordinates of the current position, and 6 blocks of the current program. When the program is running, the displayed blocks are updated dynamically. The cursor is located at the currently running block.

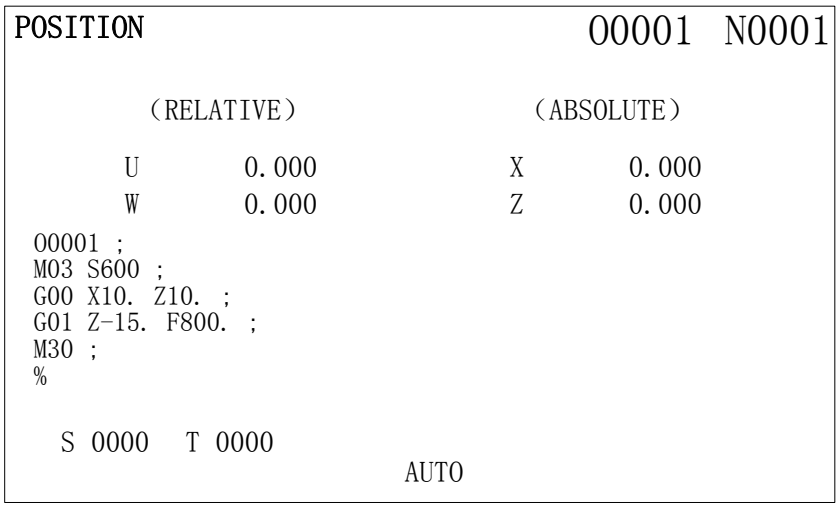


Figure III-5

2.4.2 Program screen

To enter the program screen, press the key PRGRM. In the operation modes other than EDIT, there are four pages under the Program screen, i.e. Program Content, Program State, Program Directory and U Disk, etc., which can be looked up by pressing repeatedly the key PRGRM. To look up the Program Content page in EDIT mode, press ↑ PAGE and PAGE ↓ to display all blocks of the current program.

Program Content page

In the Program Content page, it displays the program contents including the current block. When it is impossible to show all programs in one page, you can look up the program contents backward or forward by pressing 

or 

in EDIT mode.

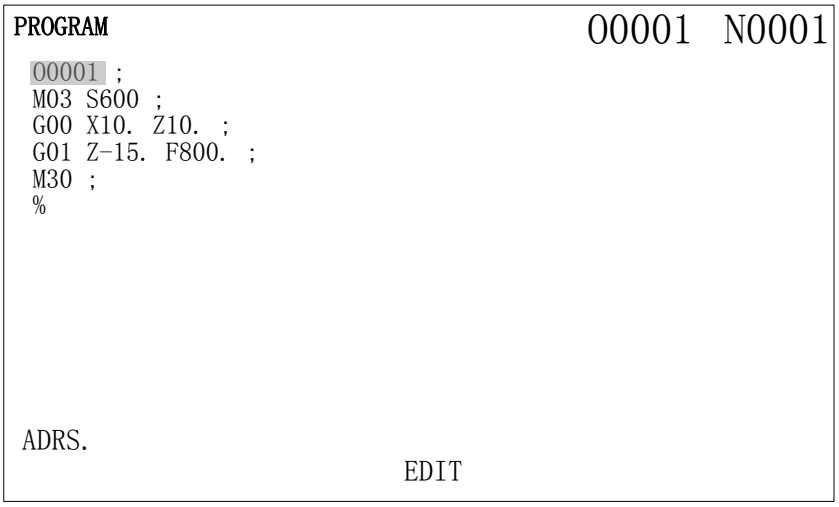


Figure III-6

Program State page

In the Program State page, it shows the state of such current commands as G, M, S, T, and F, and displays the contents of current block in MDI mode.

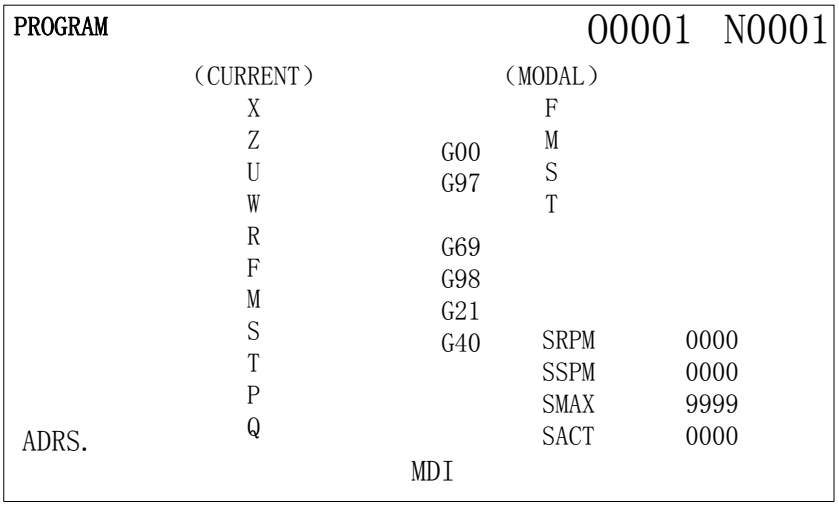


Figure III-7

Program Directory page

Contents displayed in the Program Directory page:

1 SYSTEM EDITION: to show the current software version no. of the CNC system.

2 PROGRAM NO.USED and FREE: to show the number of saved programs (incl. subprograms) and the

remained number of programs to be saved in the CNC system.

3 MEMORY AREA USED and FREE: to display the memory capacity and residual memory capacity in the CNC system.

4 PROGRAM LIBRARY: to show the number of the saved programs according to the sequence as the programs saved. When one page is not enough to show, press once again the key SHIFT to change to other page.

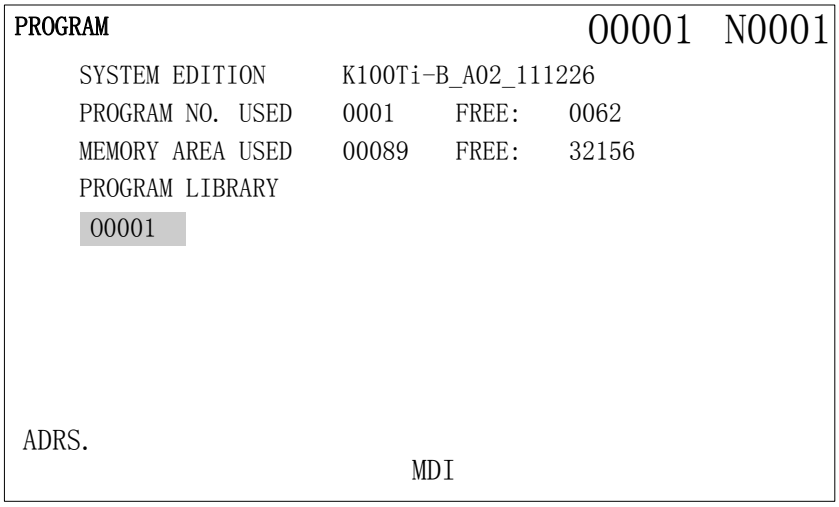


Figure III-8

U-Disk screen

In the U-Disk screen, it shows the names and sizes of programs saved in the U disk currently in use. The display screen is as follows in MDI mode:

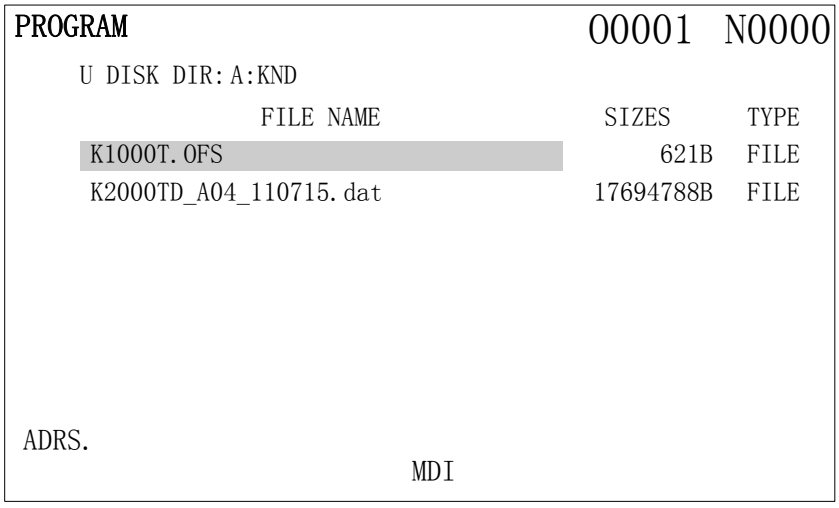


Figure III-9

2.4.3 Tool compensation/measurement display

OFSET key is a compound key. Pressing OFSET key from other screen will enter into the Tool

Compensation/Measurement screen, pressing 

again will enter into the Measurement/Tool Compensation screen.

Tool Offset

There are 8 offset numbers (No.000~No.008) available for each Tool Offset display screen. The display screen is as shown in Figure III-10:

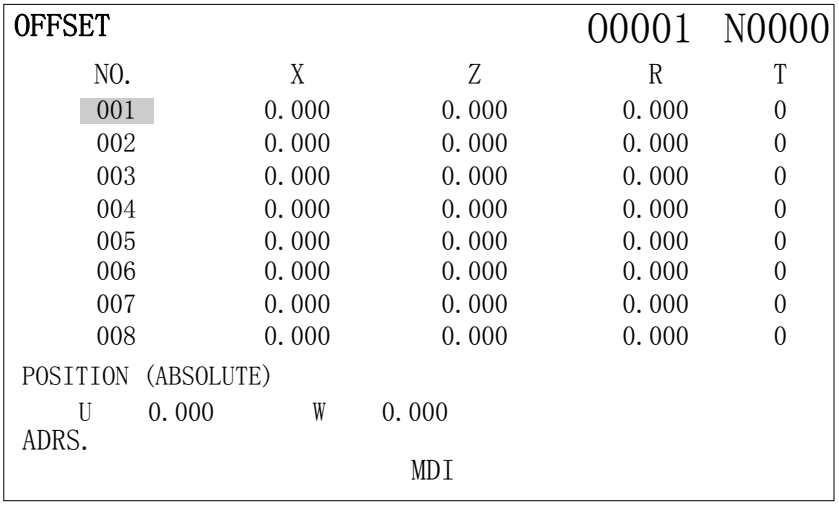


Figure III-10

Where, R means tool nose radius, and T means the imaginary tool nose direction.

Measurement

Pressing 

, 

or 

in the Tool Compensation screen can enter into the Measurement screen.

The display screen is as shown in Figure III-11:

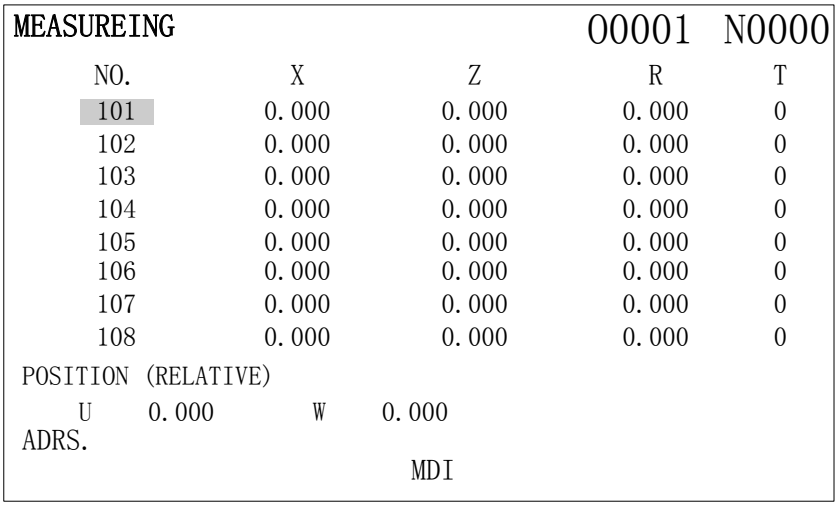


Figure III-11

Where, R means tool nose radius, and T means the imaginary tool nose direction.

Modification and setting of tool compensation

The setting of tool compensation can be divided into absolute entry and increment entry.

1 Press OFSET to show the Tool Compensation screen, press ↑ PAGE or PAGE ↓ or press OFSET in repeated way to select a desired page. It displays in two pages:

2 Left upper corner in the Offset screen: Page 1: Tool Compensation.

3 Page 2: Measurement.

4 Move the cursor to the offset no. to be input.

5 For absolute entry, press address key X or Z and numerical keys (the decimal point must be entered).

6 For increment entry, press address key U or W and numerical keys (the decimal point must be entered).

7 Press INSRT key, the compensation is entered and displayed on the LCD.

8 When the OFMD2 of parameter P042 is set as 1, the tool compensation can be input only by direct measurement. That is, in the first page (Tool Compensation screen) of the Tool Compensation screen, only address U/W can be input, and in the second page (Measurement screen), only address X/Z can be input.

NOTE

1 At the corresponding Tool Offset number of the Tool Compensation screen for Tool Offset, you can directly input the tool offset using address X or Z for absolute entry, or you can change the machining amount by modifying the tool compensation for increment entry using address U or W. The value input at the corresponding Tool Offset number in the Measurement screen of the second page of Tool Offset is the measured value; the value input at X axis is the diameter at the trial cutting point; the value input at Z axis is the distance from the trial cutting point to the origin of the work coordinate that has been established.

2 In the Tool Compensation screen, below the data display line, it displays the position coordinates; pressing SHIFT can shift the display between the relative and absolute coordinates.

3 3. In Auto mode, when the compensation amount is changed, the new compensation amount can't be valid immediately, instead it can be valid only after running the T code that specifies its offset no.

2.4.4 Parameter displaying, searching and setting

By parameter setting, it is possible to adjust the behavior of the driver and the machine tool, etc. As for the detailed meanings of parameters, refer to appendix A.

Parameter displaying

Pressing the button PARAM will enter into the Parameter screen, pressing ↑ PAGE or PAGE ↓ can shift

every parameter pages, as shown in Figure III-12:

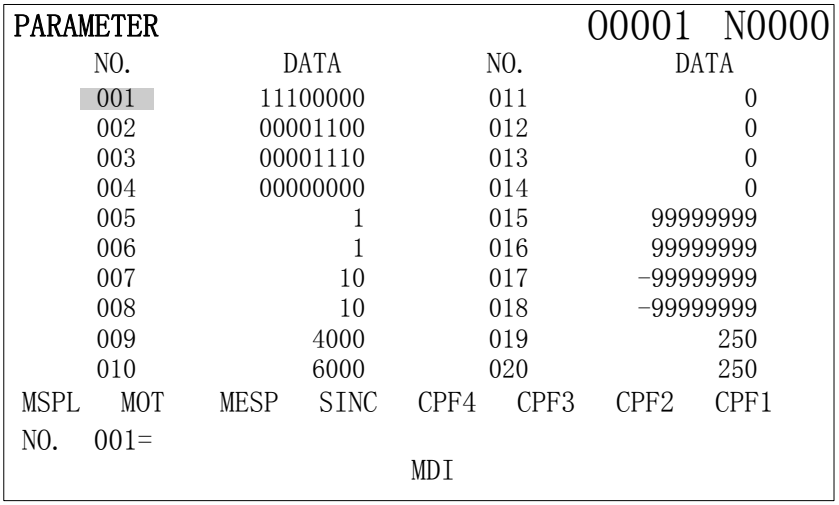


Figure III-12

In the Parameter screen, at the lower part of the LCD, there is a line for displaying the details of parameters, which displays the details of the parameter where the cursor currently is located.

1 Bit parameter

Parameters including № 001~004 and 041~044 are bit parameters, the leftmost side is the highest bit, sequentially it is BIT7~0. The abbreviations of English meanings of all bits of the parameter are displayed.

2 Data parameters

The line displaying the detailed contents of a parameter. For example, if the cursor is at №005, it will display: X axis command multiplier.

Parameter searching

1 Searching in sequence:

Each page of the Parameter screen displays 20 parameters. Pressing 

or 

based on the parameter no. will display the page where the parameter to be set is, pressing  or  , the cursor will move up and down continuously; the cursor will move up and down continuously; The name and/or meaning of the current parameter is shown on the status line, by these keys, you can find the desired parameter.

2 Direct searching

This operation can be done in EDIT/MDI/AUTO mode. First press 

key, then type in the parameter no. to be searched, and then press 

key, now the cursor will automatically jump to the parameter you want to get.

Parameter setting

1 Press 

to enter into the TEST screen, the Parameter Setting is on, press  key to enter into MDI mode;

2 Press 

key to select the Parameter screen;

3 Get the parameter to be modified via the parameter no.;

4 Type in the parameter by numerical keys;

5 Press, 

the parameter value is entered and displayed.

NOTE

For some parameters, set value will not become valid unless restarted (in this case, P/S 000 alarm will occur).

2.4.5 Diagnosis screen

All the state of DI/DO signals between the CNC system and the machine, that of the signal transmitted between the CNC system and PC, that of the internal data of PC and that of the interior of CNC system, etc. can be displayed by the Diagnosis screen.

The Diagnosis screen has one page and displays the diagnosis data; by certain operation, the same diagnosis no. can also display other diagnosis data.

Standard Diagnosis data

Press 

key to select the Diagnosis screen.

Selective Diagnosis data

In the Diagnosis screen, at the lower part of the LCD, there are 3 lines for displaying the details of diagnosis, which display the details of the diagnosis no. where the cursor currently is. As for the displayed contents, please refer to the appendix B. Example:

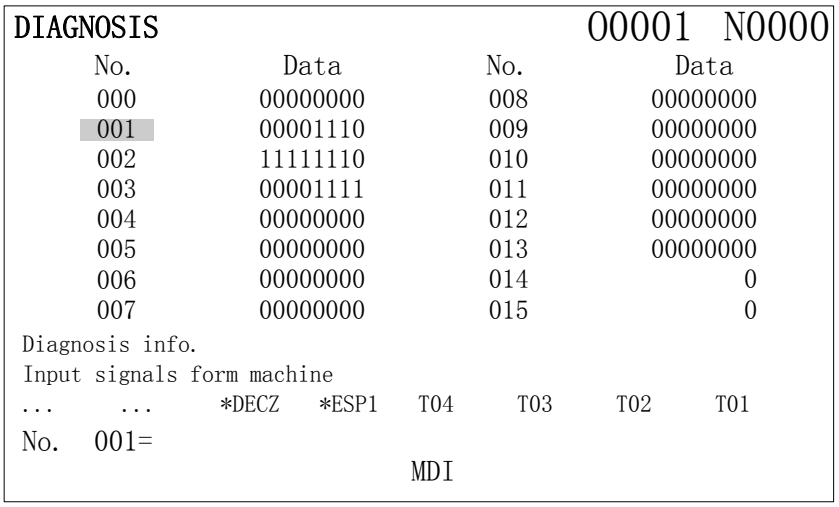


Figure III-13

If you want to know the current tool no. in the tool rest, press DGNOS key and enter into the Diagnosis screen (as shown in Figure III-13). Move the cursor to no. 001, look up the last 4 bits of no.001 diagnosis. If the first bit at right of No.001 becomes 0, namely, no.001 diagnosis information is 00111110, you can confirm that the current tool is no.1 tool. Therefore, any problem in the electric system of the machine can be judged by help of the diagnosis information, which eases the repairing.

2.4.6 Alarm screen

When an alarm occurs, it displays in flickering way "Alarm" at the lowest line of the LCD. The Alarm screen can display the alarm no. and alarm contents. As for the meanings of the alarm no., please refer to the appendix C. The Alarm screen is as shown in Figure III-14:

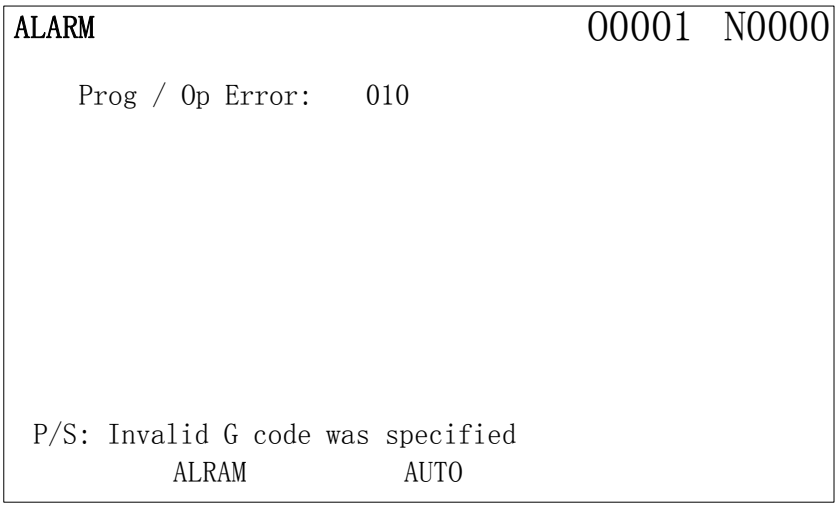


Figure III-14

In the Alarm screen, there is a line for displays the details of the P/S alarm no. at the lower part of the LCD. The details of other alarms (overtravel alarm, driver alarm and CNC alarm) will be directly displayed in the middle of the LCD.

NOTE

1 Normally when an alarm occurs, the display will automatically shift to the Alarm screen to show the contents of the alarm.

2 When there is no alarm and the system is under pause state, at the location formerly for displaying 'alarm' in flickering way at the lower part of the LCD, ‘Pause’ will be displayed in flickering way.

3 Way for canceling alarm: when a p/s alarm occurs, pressing RESET can cancel the alarm.

2.4.7 Graph screen

The Graph screen covers two windows, i.e. Graph Setting and Graph Display, etc., you can look up by pressing repeatedly GRAPH key, or pressing ↑ PAGE or PAGE ↓ key.

Graphic Parameter Setting screen

In the Graph Setting screen, you can select the coordinates system, scale and range for the graphic display (as shown in Figure III-15).

```
GRAPHIC                                    00001  N0000
                    GRAPHIC PARAMETER
        AXIS SEL.     =          0  (XZ:0  ZX:1)
        SCALERATE     =          0  (Auto Cal.)
        X CENTER      =      0.000  (Auto Cal.)
        Z CENTER      =      0.000  (Auto Cal.)
        X MAX. ABS    =      0.000
        Z MAX. ABS    =      0.000
        X MIN. ABS    =      0.000
        Z MIN. ABS    =      0.000

NO.   001=
                                MDI
```

Figure III-15

Graph Display screen

In the Graph Display screen, it displays the paths in the parameter range of the Graph Setting screen (taking the absolute coordinates as reference) (as shown in Figure III-16)

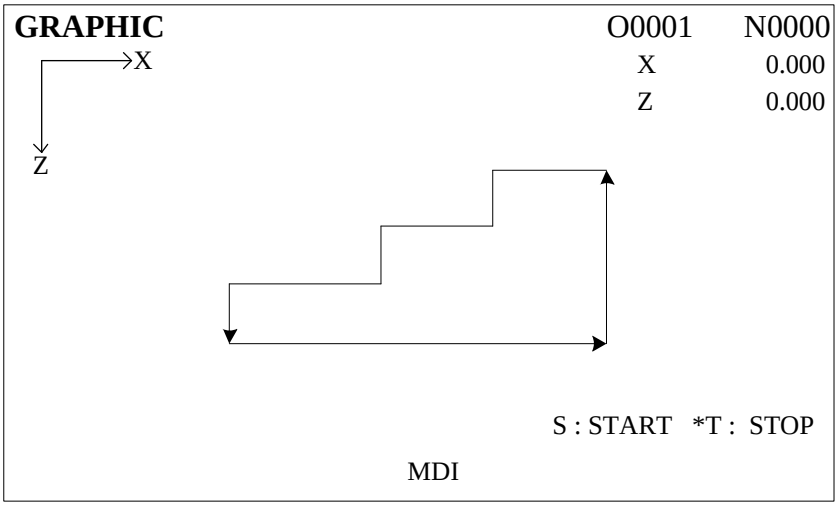


Figure III-16

NOTE

As for the specific using of the graphic functions, refer to section 3.9.

2.4.8 Test screen

The Test screen is as shown in Figure III-17:

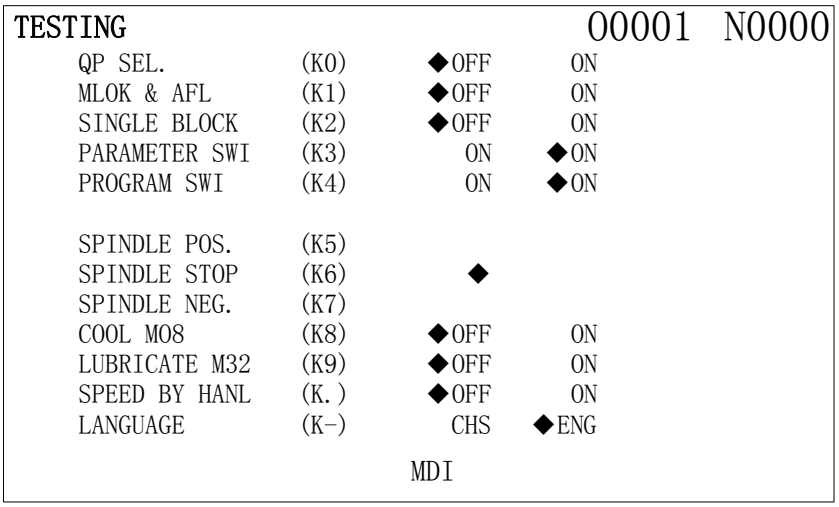


Figure III-17

QP SEL.(Inner/outer chuck selection, key 0)

QP SEL allows a user to select different type of chucks. This selection will maintain after power-off. Press 0 repeatedly to switch between inner and outer chuck.

MLOK&AFL (Test Run, key 1)

When the Test turns on, the machine will not move when a program is started, M, S, T will not output, and the feedrate will be that for dry running (note 2), but the position coordinates change. This is for

checking the programs. Pressing repeatedly [1] key can change the on/off of Test Run.

NOTE

1 When the Test or Single Block switch is 1, it displays flickeringly "TESTING" at the status line.

2 At cutting feed, the rate shall be the max. manual feed rate when pressing the Manual Rapid Feed button (1260 mm/min). Otherwise, it will be the manual feedrate.

SINGLE BLOCK (key [2])

When the Single Block switch is on, it will stop after executing one block. If pressing [ ] again, the next block will be performed, and then the machine stops. Pressing repeatedly [2] key can change the on/off of Single Block.

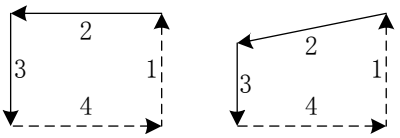
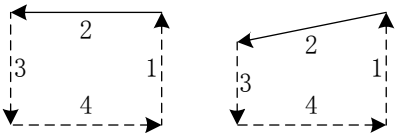
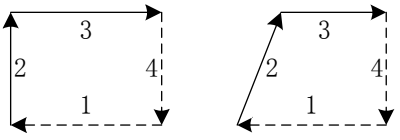
NOTE

1 In G28, even if it is at an intermediate point, the stopping for Single Block will still be performed.

2 2.The block of M98 P__; and M99 can't be stopped in Single Block way. But if any other address other than N, P exists in M98 and M99, they can be stopped in Single Block way.

When Single Block is On, and when executing canned cycle G90, G92 and G94, the following (as shown in Table III-4) will happen.

Table III-4

G code	Tool path (Back tool rest)	Description
G90		Steps from 1 to 4 compose one cycle. Tool stops after step 4.
G92		Steps from 1 to 4 compose one cycle. Tool stops after step 4.
G94		Steps from 1 to 4 compose one cycle. Tool stops after step 4.

PARAMETER SWI & PROGRAM SWI

1 Parameter switch (key 3): parameters can be set only when the parameter switch is on. Pressing repeatedly [3] will change on/off the parameter switch.

2 Program switch (key 4): programs can be edited only when the program switch is on. Pressing repeatedly [4] will change on/off the program switch.

Miscellaneous functions

Selecting JOG mode (Manual, Home, Step, Handwheel) can control the outputting and turn off miscellaneous functions of the machine by key 5~9 in the Test screen. In addition, you can know the current state of the output of miscellaneous functions in the system.

1 SPINDLE POS.(spindle CW, key )

2 SPINDLE STOP(spindle stop, key )

3 SPINDL NEG.(spindle CCW, key )

4 COOL M08 (coolant, key 8). Pressing repeatedly  key can change the on/off of coolant.

5 LUBRICATE M32 (Lube, key )

Auto Handwheel Function (key )

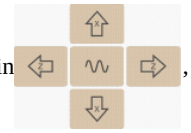
Press key  repeatedly will choose this function on/off, See “3.6 Controlling Feed Rate by Handwheel” in this part.

Language selection

Press key  repeatedly , the text of the Interface is switched between Chinese and English。

3 MANUAL OPERATION

3.1 Manual Continuous Feed

In JOG mode, pressing  or  will feed in  , once the key is released, the axis motion will stop; pressing  or  will feed in Z axis + direction or Z axis - direction, once the key is released, the axis motion will stop. You can press simultaneously X axis and Z axis to realize two axes moving simultaneously.

In JOG mode, if pressing  and direction key, tool can rapid traverse at the direction of the selected axis.

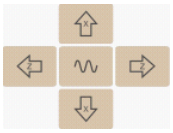
In the Position page, pressing  or  can select manual motion feedrate (0~1260mm/min).

NOTE

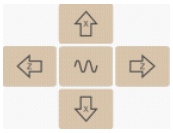
The feed rate, time constant and acceleration /deceleration mode at manual rapid traverse are the same as the rapid traverse (G00) commanded by program.

3.2 Step Feed

In the Step mode, select movement amount (in the Position page, select override via  or  , there are 4 selections for increment: 0.001, 0.01, 0.1, 1.0 mm).

Press  Pressing once, it will move once. Parameter P001 SINC can set to mask step 0.1 and 1.0.

3.3 Handwheel Feed

In Handwheel mode, selecting  (in relative coordinate and absolute coordinate position page, pressing X or Z manual axis key, the selected hand wheel axis coordinate address word will flicker) and pressing  or  , the hand wheel will CW or CCW. There are 3 selections for

increments: 0.001, 0.01, 0.1mm (in the absolute or increment position page at down left corner in the display).

NOTE

1 Step mode and hand wheel mode are using the same key. HPG of PA 04 specifies the selection.

2 Hand wheel's speed must be less than 5 rev/s. If speed is more than that, even if hand wheel ends rotation, axis cannot stop immediately, which causes disagreement between scale and the movement amount.

3 Setting parameter P0001 is equal to 1 can shield 0.1 gear.

3.4 Spindle Feed

3.4.1 Spindle CW

 : In MANUAL or STEP mode, the spindle starts to rotate clockwise when the key is pressed.

3.4.2 Spindle CCW

 : In MANUAL or STEP mode, the spindle starts to rotate counter-clockwise when the key is pressed.

3.4.3 Spindle stop

 ; In MANUAL or STEP mode, the spindle stops when the key is pressed.

3.4.4 Spindle jog

 : In MANUAL or STEP mode, the spindle rotates clockwise when the key is pressed and the spindle stops when the key is released.

3.5 Other Manual Miscellaneous Operation

3.5.1 Manual tool change

 : In the MANUAL/STEP mode, pressing the key, the tool rest will rotate and the next tool is used. (Refer to the instructions from the machine builder)

3.5.2 Coolant control

 : In the MANUAL/STEP mode, pressing the key and the Lock button, the coolant will switch 'on→off→on...'.

3.5.3 Adjusting rates

In current or Relative Position page, it is possible to select machine feed rate:

: In MANUAL or AUTO mode, feedrate increases.

: In MANUAL or AUTO mode, feedrate decreases.

: Handwheel/Step increment increases.

: Handwheel/Step increment decreases.

: Home rate increases, rapid traverse override increases.

: Home rate decreases, rapid traverse override decreases.

: Spindle speed increases.

: Spindle speed decreases.

NOTE

1 Rapid traverse override and spindle speed override display

In the Position page, the traverse override and spindle speed override are displayed at same position and shifted by 

key. If having no analog spindle function, the system does not display spindle speed override.

2 Spindle override increment increase or decrease is not enabled unless the spindle analog function is enabled.

Increase: pressing 

once, the spindle override will increase according to the following sequence: 50%→60%→70%→80%→90%→100%→110%→120%

Decrease: pressing 

once, the spindle override will decrease according to the following sequence: 120%→110%→100%→90%→80%→70%→60%→50%

As for a machine with a frequency conversion motor controlled spindle, spindle speed (rpm) is directly commanded via S+4-digit.

As for a machine with a multiple-speed spindle motor, the spindle gears (S00~S04) are controlled via S＋2-digit.

3 No matter for a machine with a frequency conversion motor controlled spindle or a machine with a multiple-speed spindle motor, it is necessary to input spindle speed (S××××) or gear signal (S××) in MDI when restarting after machine tool power OFF.

After pressing Start button, the spindle will run only by starting it in Manual/Step mode.

4 During tool change,  is disabled. Pressing 

or EMERGENCY STOP can turn off tool rest CW /CCW output, and end the tool change. After starting in MANUAL mode, when the mode is changed, the output keeps invariable. But in AUTO mode, executing corresponding M code will turn off corresponding output. After executing corresponding M code in AUTO mode,

5 At the moment of power ON, in MANUAL mode, by pressing  ,  ,  , the spindle usually will not rotate. But by executing a speed in MDI mode: inputting a speed directly for a machine using a frequency conversion motor, for example S500, or inputting a speed gear for a machine using a multiple speed spindle motor, for example S01, the spindle can be started by returning to MANUAL mode after that. the corresponding output can be turned off by pressing the corresponding key. At emergency stop, the spindle, coolant, tool change output are turned off.

4 AUTOMATIC OPERATION

4.1 Selection of Running Program

Searching

1 Select EDIT or AUTO mode;

2 Press 

, enter into Program Content page;

3 Press address key 

and type in the program number;

4 Press 
, on the display page it displays the searched program, if the program does not exist, the CNC alarms.

NOTE

If the program does not exist, (in EDIT mode and program switch is on) replace  by 

in step 4, CNC will create a new program.

Scanning

1 Select EDIT or AUTO mode;

2 Press 

, enter into the Program Display page;

3 Press address key 

;

4 Press 
to display the next program. If current program is the last program, the first program will be regarded as the next program.

4.2 Start Automatic Operation

1 Press 
to select Auto mode;

2 Press 
to start program, and the program runs automatically.

NOTE

Program starts from the line where the cursor is, check whether the cursor is at the required block before running the program. It is necessary to place the cursor at the start line to run program from start. Method: in AUTO or EDIT mode, press  ,  and  , the cursor automatically skips to the program start line.

4.3 Stop Automatic Operation

Automatic operation can be stopped using one of two methods: Set a stop command, or press a key on the machine operator's panel.

Program Pause (M00)

Automatic operation is stopped after a block containing M00 is performed. When the program is stopped, all existing modal information remains unchanged. The automatic operation can be restarted by actuating the cycle operation.

Program End (M30)

1 Main program ends.
2 Memory operation is terminated and the reset state is entered.
3 Returns to the start of the program

Program End (M02)

1 The output of M codes remains unchanged, others are the same as M30.
2 This command must be specified in a separate block.

Dwell

When the three-position switch on the operator's panel is in the middle during automatic operation, the tool decelerates to stop. The machine state is as follows:

- The feed is stopped after deceleration if the machine is moving.
- Dwell is stopped if the dwell is being performed (G04).

In AUTO mode, the spindle will stop when the three-position switch is in the right; the spindle will continue to run as soon as the three-position switch returns to the middle position.

Reset

Automatic operation can be stopped and the system can be made to the reset state by using  key on the LCD/MDI panel or by an external reset signal. When reset operation is applied to the system during a tool movement, the motion slows and then stops.

4.4 Auto Run from Any Block

1 In the EDIT mode, press  to enter into Program screen, press repeatedly to select Program

Content page;

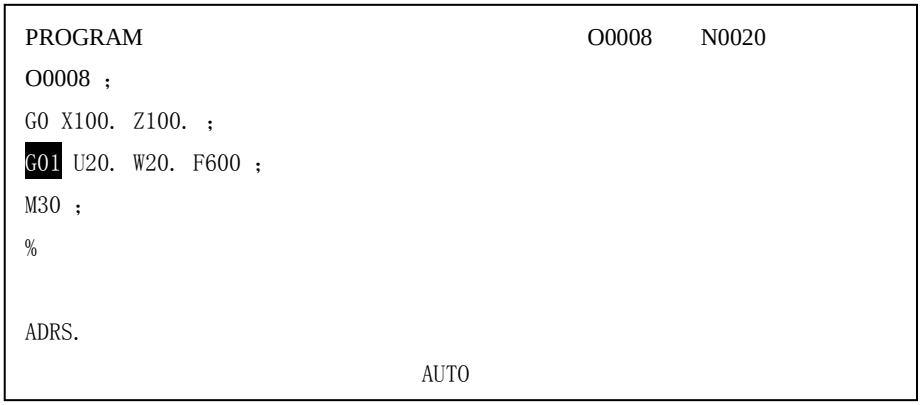


Figure III-18

2 Move the cursor to the block where the operation is to start, for example, if you want to start from the fourth line, place the cursor at start of the fourth line;

3 If the modal command (G, M, T, F) of the block where the cursor currently is located is default, and is not identical with the modal of the block to be run, the next procedure can be proceeded only after executing corresponding modal function;

4 Press  to enter into Auto mode, and press  to start program running.

4.5 Pause or Feed Hold in AUTO Mode

In AUTO mode, when the command dwells or the Pause 3-position knob is in the state of feed hold, the program running can continue after restoring the Pause knob by pressing  key.

4.6 Feed or Traverse Rate Adjustment in AUTO mode

In AUTO mode, the run speed can be changed by adjusting the feedrate and rapid traverse override without changing the program and the parameter set speed.

4.6.1 Adjusting feed override

1 Press 

, the feed override increases one step, until 150％;

2 Press 

, the feed override reduces one step, until 0.

Sixteen steps of real-time adjustment in feed override can be achieved by above way.

NOTE

1 Feed override adjustment will use the value specified by F in the program;

2 2. Actual feed rate＝F specified value ×feed override

4.6.2 Adjusting traverse override

1 Press 

, the feed override increases one step, unless 100％;

2 Press 

, the feed override reduces one step, unless F0.

Four steps of real-time adjustment in traverse override (FO, 25%, 50%, 100%) can be achieved by above way.

NOTE

1 No.009, No.010 set X, Z axis rapid traverse rate;

2 X axis actual rapid traverse override ＝No.009 specified value ×traverse override

3 Z axis actual rapid traverse override ＝No.010 specified value ×traverse override

4.7 Spindle Speed Adjustment in AUTO mode

In AUTO mode, when selecting analog voltage output to control spindle speed, the spindle speed can be adjusted.

Method: setting SANG of NO.004 to 1, select analog spindle function, in the Position page (relative coordinates or absolute coordinates)。

Press 

, feed override will increase one gear, until 120%;

Press 

, feed override will reduce one gear, until 50%.

It is possible to realize real time adjustment in spindle override (50%~120%) by changing spindle speed via adjusting spindle override change.

NOTE

1 Actual output analog voltage = calculated analog voltage according to parameter x spindle override.

2 NO.031, 032 is set to 9999.

4.8 Coolant Control in AUTO mode

In AUTO mode,  in machine tool control area is enabled, pressing the key repeatedly can switch on /off coolant.

5 MDI OPERATION

In MDI mode, you can set parameters as well as input and execute command words.

5.1 Inputting MDI Command Words

Select MDI mode and enter into the Program State screen, input a block G50 X50 Z100, the steps are:

1 Press  to enter into MDI mode;

2 Press 

to enter Program State screen, and repeat pressing the key if necessary:

```
PROGRAM                                  O0001    N0063
        （CURRENT）                （MODAL）
              X                          F
              Z                G00       M
              U                G97       S
              W                          T
              R                G69
              F                G98
              M                G21
              S                G40   SRPM    0000
              T                      SSPM    0000
              P                      SMAX    9999
              Q                      SACT    0000
ADRS.
                                MDI
```

Figure III-19

3 Type in sequentially address key 

, numerical key 

and 

, then press key 

, and the screen will display as follows:

```
PROGRAM                                  O0001    N0063
        （CURRENT）                （MODAL）
      G50   X                          F
              Z                G00       M
              U                G97       S
              W                          T
              R                G69
              F                G98
              M                G21
              S                G40   SRPM    0000
              T                      SSPM    0000
              P                      SMAX    9999
              Q                      SACT    0000
ADRS.
                                MDI
```

Figure III-20

4 4. Type in sequentially address key Z, numerical key 5 and 0, and INSRT key;

5 5. Type in sequentially address key X, numerical key 1, 0, 0 and INSRT key; Then the screen displays as follows:

```
PROGRAM                                          O0001     N0063
          （CURRENT）                （MODAL）
     G50   X      79.000                F
           Z      20.000        G00     M
           U                    G97     S
           W                            T
           R                    G69
           F                    G98
           M                    G21
           S                    G40    SRPM     0000
           T                           SSPM     0000
           P                           SMAX     9999
           Q                           SACT     0000
ADRS.
                                MDI
```

Figure III-21

5.2 Running and Stopping MDI Command Words

After inputting the command words, pressing [cycle start] will execute MDI command words. You can stop the MDI command words during the running by rotating the EMERGENCY STOP (ES) button.

NOTE

The input block will not be saved, and will disappear after execution; only one block can be input once.

5.3 Modifying MDI Command

In MDI mode and Program State screen, before executing the input data, if any error exists in the block inputting, you can re-input the correct value to replace the erroneous content. For example, in section3.6.1, Z50 is input incorrectly, to input the correct Z100, you can type in address key Z, numerical keys 1、0, 0 and INSRT key to replace the incorrectly input Z50. After that, the screen will display as follows:

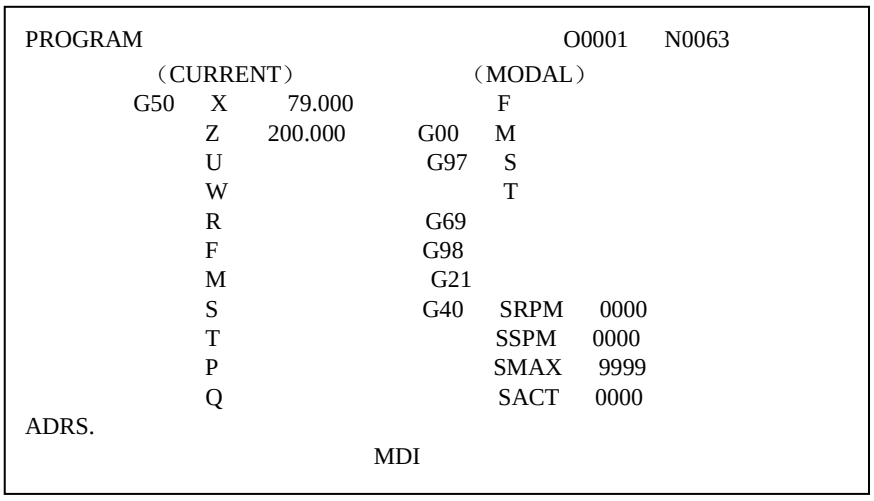


Figure III-22

6 CONTROL SPEED BY HANDWHEEL

This function is enabled when turning on the soft switch of “Auto Handwheel” in Debug screen and current mode is Auto or MDI mode.

Function description

The tool path is controlled by the machining program under program running, and its movement speed is controlled by the input from the hand wheel. The feed rate is proportional to the absolute value of hand wheel input. And the feed rate specified by the program is invalid until this function is canceled.

“Auto”, “Handwheel” or “MDI”, “Handwheel” will be displayed on the status bar.

When this function is enabled, the machine movement distance for one hand wheel pulse can be adjusted in “Auto” or “MDI” mode, and the pulse equivalences are displayed in “Position” screen.

NOTE

When this function is enabled, the feed per revolution and the feed rate for machining thread is also controlled by the hand wheel.

7 RETURN REFERENCE POSITION

7.1 Program Reference Position Returning

7.1.1 Program reference position

When a part is clamped on a machine, the absolute coordinates of the current position of the tool are set by G50 depending upon the relative position between the tool and the part, so a work coordinates system is established in the CNC system. The current position of the tool is called the program zero, after executing the program, the zeroing (homing) will return to this position.

In manual Program Homing, same as manual returning to reference point, it is possible to rapidly return manually to the tool-starting point set by G50.

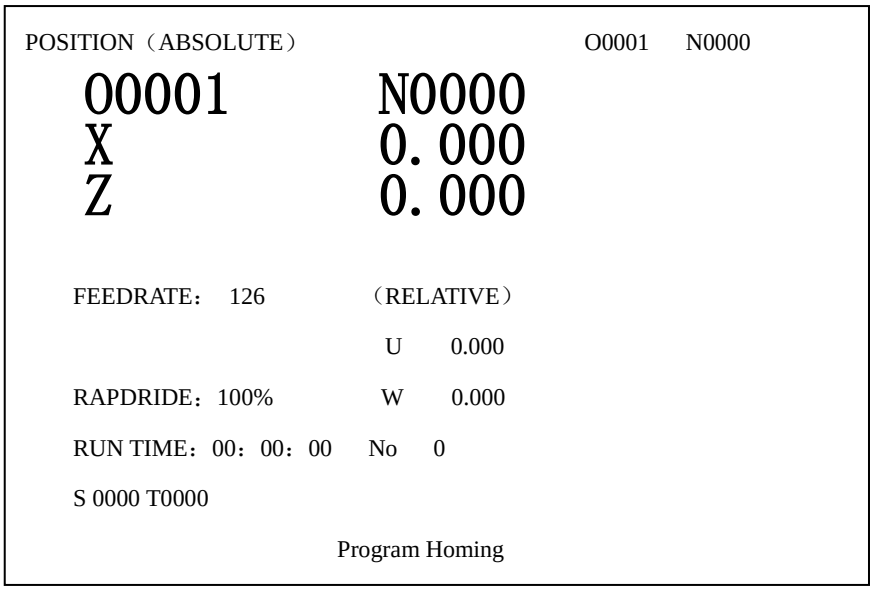
Program zero memory: after a program starts, the machine tool position where executing the first G50 block will be memorized automatically. The following G50 code will not be memorized.

Once the program zero is memorized, it will be maintained unless a new zero memory occurs. That is, if program zero A is memorized when executing program A, then zero A is always memorized when executing program B (supposing there is no G50 in B) even if the program B is finished.

Use: it allows returning to the machining start rapidly in a manual way after the program stops halfway. The tool offset compensation is automatically cancelled. ALARM(№90) will occur if Program Homing is carried out without any memorized zero.

7.1.2 Operations

1 Hold down simultaneously the address  and  key, on the lower right corner, it displays “Program Homing” mode as follows:



2 Press the directional key for X and Z axis to zero; the address flickers after returning to the program

zero.

NOTE

Under mechanical homing, it displays “HOME”, under Program homing, it displays “Program Homing”.

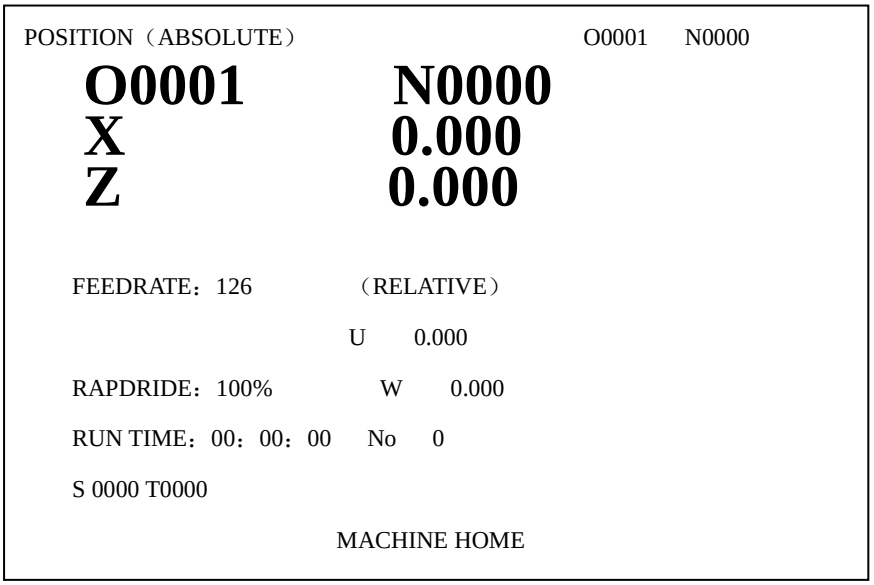
7.2 Mechanical Reference Position Returning

7.2.1 Mechanical reference position

A machine coordinates system is a reference coordinates system for the CNC system to calculate coordinates, is an inherent coordinates system of the machine. The origin of the machine coordinates system is called mechanical zero (or mechanical reference point). The mechanical zero is decided by the zero switches or home switches on the machine, which are normally mounted at the points with the maximum strokes in the positive direction of X and Z axis respectively.

7.2.2 Operations

1 Press  to enter into Mechanical Homing mode, the lowest line in the display screen will show “HOME” as follows:



2 Press  or  to return to the mechanical zero of X or Z axis;

3 The address of the axis that has returned to the reference point flickers in the Position screen; after the axis moves out, the address stops flickering.

NOTE

1 When parameter P003 ZNIK is set as 1, the moving axis is self-protected, and the axis can stop after automatically moving to the mechanical zero. If it is necessary to stop halfway, press 

key.

2 2. The parameter P004 MZRZ and MZRX select which direction(+ or -) that the axis manually returns to the reference point is valid.

8 PROGRAM STORAGE AND EDITING

In EDIT mode, you may set, select, modify, copy and delete program, also realize bi-directional communication between CNC and PC.

In order to avoid that a program is accidentally modified or deleted, K100Ti-B is provided with a program switch. Before editing a program, the program switch must be on.

8.1 Creating New Program

8.1.1 Block number

In EDIT mode and in the Test page, set the program switch to be on, press in sequence  , program number, and  , it will display the interface for new program as shown in Figure III-23: the program number is 02.

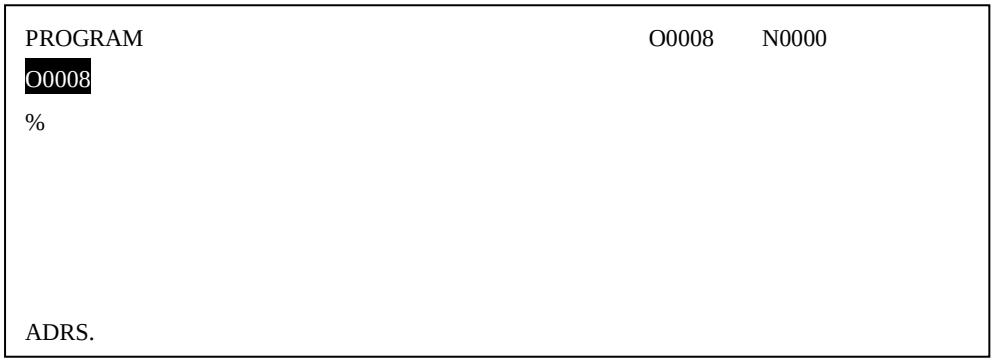
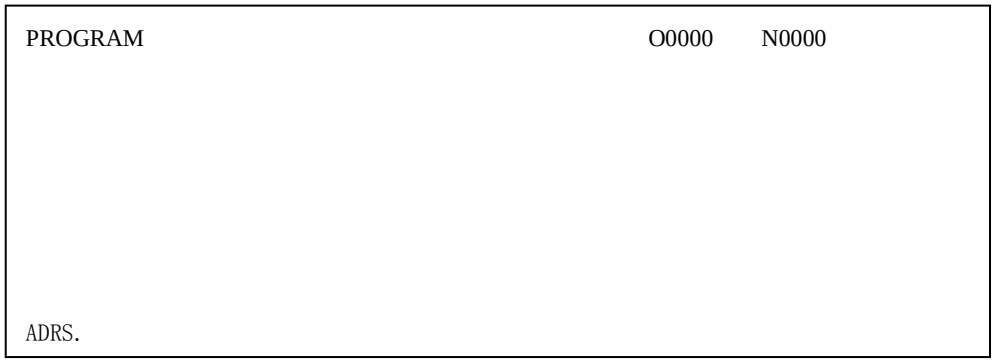


Figure III-23

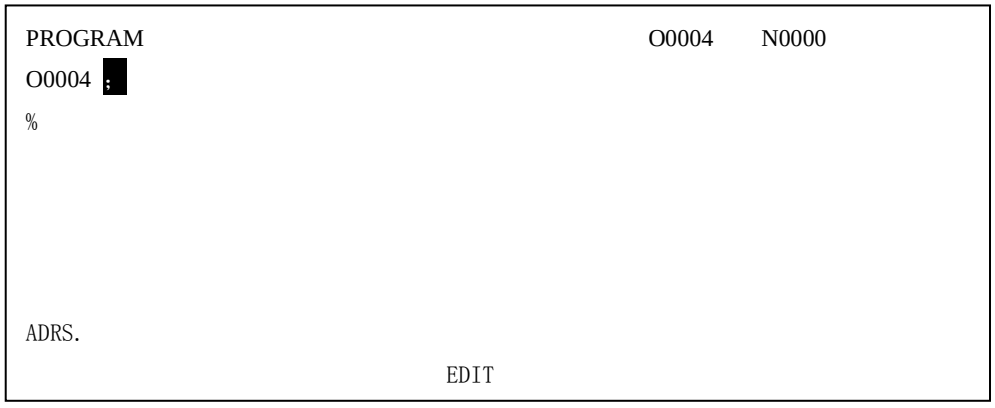
8.1.2 Inputting program content

1 Press  to enter into Edit mode;

2 Press  to enter into Program page, press  repeatedly to select the display screen for program contents.



3 Press O, 0, 0, 0, 2, and EOB to create a new program (take O0002 program as an example)



4 Input programs according to the written part program. The display shows the character once it is input.

When the inputting of a block finishes, press EOB to end, the cursor will go to a new line.

8.1.3 Searching characters

A character can be searched by merely moving the cursor through the text (scanning) by word searching or by address searching.

Word scanning

The cursor moves forward word by word on the screen; the cursor is displayed below the address character of a selected word.

1 Pressing the ← button

The cursor moves forward word by word on the screen; the cursor is displayed at a selected word.

2 Pressing the → button

The cursor moves backward word by word on the screen; the cursor is displayed at a selected word.

3 Holding down the ↓ to scan words continuously

The cursor moves downwards line by line on the screen; the cursor is displayed at a selected word.

4 Holding down the ↑ to scan words continuously

The cursor moves upwards line by line on the screen; the cursor is displayed at a selected word.

5 Pressing the page key PAGE ↓ to display the next page and search the first word of the page.

6 Pressing the page key ↑ PAGE to display the previous page and search the first word of the page.

Word searching

A specified words can be searched from the current position in forward or backward direction.

1 Type in address S.

2 Type in '1' '2'.

NOTE

1 S12 cannot be searched if only S1 is typed in.

2 S09 cannot be searched by typing in only S9. To search S09, be sure to type in S09.

3 Press the ↓ key to start searching.

Procedure for searching an address

A specified address can be searched from the current position in forward direction.

1 Type in an address, such as X.

2 Press the key ↓.

Pressing the key ↑ rather than the key ↓ will execute searching in reverse direction.

Returning to a program start

- **Method 1**

Press RESET when the Program screen is selected in EDIT mode. When the cursor has returned to the start of the program, the contents of the program are displayed from its start on the LCD screen.

- **Method 2**

Execute a Program Number searching.

- **Method 3**

1 Set the mode selector switch to AUTO or EDIT.

2 Press the key PARAM to display the program.

3 Press the address key O.

4 Press ↑ key.

8.1.4 Inserting a character

Operation method and steps are:

1 Select EDIT mode, and enter into Program Content page as shown in Figure III-24.

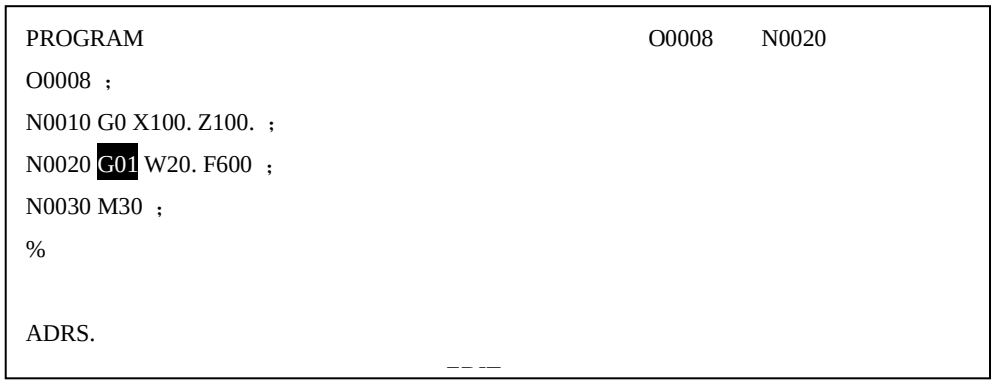


Figure III-24

2 If you want to insert Z10 after X50, move cursor to X50 as shown in Figure III-24. Input the character to be inserted (press in sequence Z, 1, 0, and press INSRT), now the character insertion is over, then the page shows as in Figure III-25.

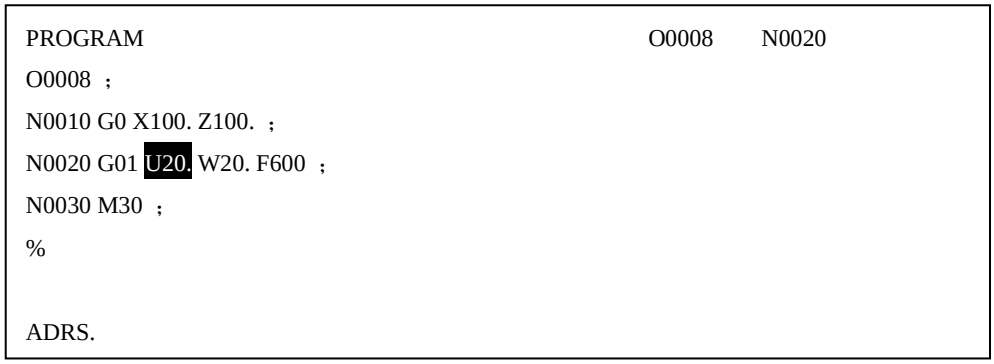


Figure III-25

8.1.5 Altering a character

1 Search or scan the character to be altered.

2 Input the address and data to be altered, press ALTER key. The input word will replace the word pointed by current cursor.

8.1.6 Deleting a character

1 Search or scan the character to be deleted.

2 Press DELET button to delete the content pointed by the current cursor.

8.2 Program and Sequence No. Searching

8.2.1 Program searching

When the memory saves several programs, a program can be searched. In general, the program that pointed by current program pointer is displayed. The program pointer can't lose even if power OFF. The desired program can be called by searching, the called program can be edited or executed. This operation is called program searching.

There are two methods for that as follows:

Searching

1 Select mode (EDIT or AUTO)

2 Press the address key  .

3 Input the desired program number.

4 Press the key  .

5 The searched program displays on the screen and the searched program number appears at the right of the screen after searching.

Scanning

1 Select mode (EDIT or AUTO)

2 Press the  button to display the program.

3 Press the address key  .

4 Press the  key. In EDIT mode, press the address key  and cursor  key to display the registered program one by one.

8.2.2 Sequence number searching

Sequence number searching is usually used to search for a sequence number in the middle of a program so that execution can be started or restarted at the block of the sequence number. Those blocks that are skipped do not affect the state of CNC. This means that the data in the skipped blocks such as coordinates and M, S, and T codes do not alter the CNC coordinates and model values. So, in the first block where execution is to be started or restarted by using a sequence number searching command, be sure to enter the desired M, S, and T codes and coordinates. A block searched by sequence number searching usually represents a point of shifting from one process to another. When a block in the middle of a process must be searched to restart execution at the block, set M, S, and T codes, coordinates, and so forth as required from the MDI after closely checking the machine tool and NC states at that point.

Procedure for sequence number searching

1 Set the mode selector switch to AUTO or EDIT.

2 Press the key PRGRM to display the program.

3 Select the program including the desired sequence number.

4 Press the address key N.

5 Input the searched sequence number.

6 Press the key ↓.

7 The searched sequence number is displayed at the top of the right LCD screen.

NOTE

1 During searching, the following checks are made :

（1） Optional block skip

（2） P/S alarm (No. 003 to 010)

2 During sequence number searching, M98Pxxxx (subprogram call) is not performed. So an alarm (P/S No. 060) occurs if an attempt is made to search for a sequence number in a subprogram called by the program currently selected.

8.3 Deleting Program

8.3.1 Deleting multiple blocks

Purpose: to delete the blocks starting from the displaying position to specified sequence number.

Example cursor current position—> N100 X100.0 M15 ;

N110 X50.0;

1 Press N and DELET, delete block N100, cursor points at N110 block.

2 Press N and L (block count) and DELET,delete downwards L blocks from current block (including N100 block).

3 Press N and — + L (block count)+ DELET, delete upwards L blocks (excluding N100 block).

8.3.2 Deleting program

Press address key O, input the program number to be deleted, press DELET. The program corresponding to the program no. registered in the memory is deleted.

8.3.3 Deleting all programs

To delete all programs registered in the memory, press address key O, input -9999 and press DELET.

8.4 Copying Program

The function allows copying the current program as another program. Operation way (take "save as O1000" as an example): with program switch on, input "O1000" in "Program Edit", press 

. Now the current program is copied to O1000.

When copying program, following alarms are checked:

1 ALARM: The program number has already used;

2 ALARM: The memory is full.

3 ALARM: The quantity number of program exceeds 63.

If system power is OFF during the copying process, the copying will be unsuccessful or only partial program will be copied.

8.5 Program Managing

8.5.1 Program directory

In the Program page, "Program Directory" can display program numbers in order of program sequence numbers.

8.5.2 Quantity of stored program

A standard product can register 63 programs a time in the memory.

8.5.3 Memory space

Memory: 32KB;

External memory: flash memory (6 partitions)

9 FLASH MEMORY

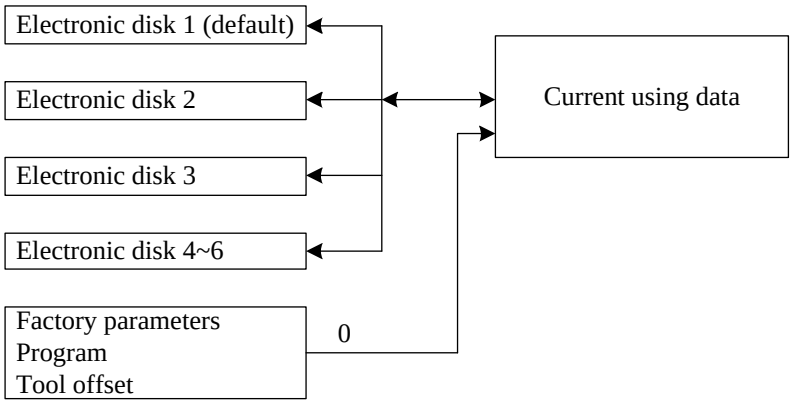
9.1 General

This system adopts flash memory. It is an nonvolatile memory.

Function

1 Backup: As battery insufficiency or other reason makes data lose, CNC system can rapidly read data from flash memory, restore program, parameters and data.

2 When the memory capacity is insufficient, the unused programs may be saved temporarily into flash memory, when use these programs, the system can read data from the flash memory at any moment.



There are six partitions in a flash memory, each includes such data as parameter, program, offset, PC parameter and so on. The system's current data can be stored in a random partition, and the system can read data from a random partition as the current data.

9.2 Read Operation

The procedure for reading data from a random partition into the working area is as follows:

1 Press INSRT + 0~6 number key and power ON

2 The following words are prompted:

'Read from flash memory, press RESET to confirm, press CAN to cancel (key in number)'.

3 Press key RESET, the disk data corresponding to the number key are read into the working area. For reading no disk, pressing CAN with power ON is equivalent to "not pressing any key".

9.3 Initialization Setting for the System

As the Program page displays messy codes or the machine tool malfunctions many times successively, you can perform system initialization setting (If data have been saved, read saved data from the disk, the system will restore quickly; only if the data are not saved, the system initialization method will be used).

Initialization method is: press 

+ 

to power ON, the system prompts ‘read flash memory’, press 

to confirm; set KND factory standard parameters, the data in program area and offset area are cleared; then the parameters are modified to the parameters for normal use (the parameters shall be backed up before shipping the machine tool for use in case of parameter loss or after initialization).You should save data in a disk after completion.

NOTE

1 The system can read disk only at the moment of power ON, and cannot read disk after that.

2 Read data after registering battery hold data into the flash memory, otherwise the read data will be incorrect.

9.4 Save Operation

The data of the system’s work area can be saved in any disk. Operation is as follows:

In Program screen, select EDIT mode, press 

, number key 1~6, press 

to save. When saving disk, it displays ‘saving flash memory’ in the state line at right down corner. When the number key is omitted, the default disk number is 1. ‘Saving ’disappears after saving the flash memory.

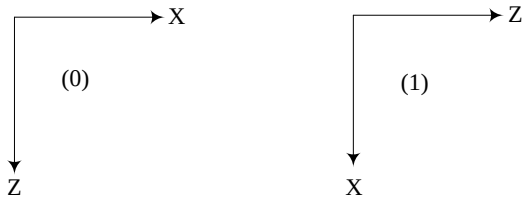
NOTE

At emergency stop, system cannot save disk

10 GRAPHIC FUNCTION

Tool path can be displayed on the LCD screen. Therefore, cutting path and machined shape can be checked on the LCD screen. The tool path can be reduced and enlarged.

Graphic Parameter allows selecting one of the two coordinates systems:



Dimension of LCD:

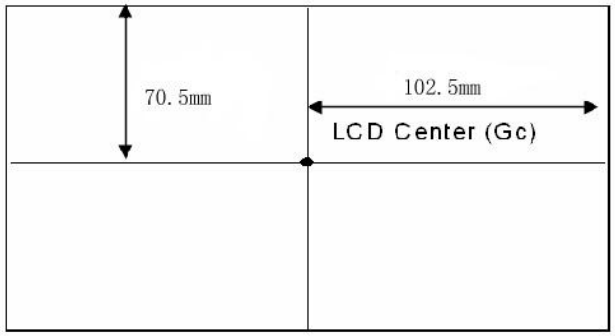


Figure III-26

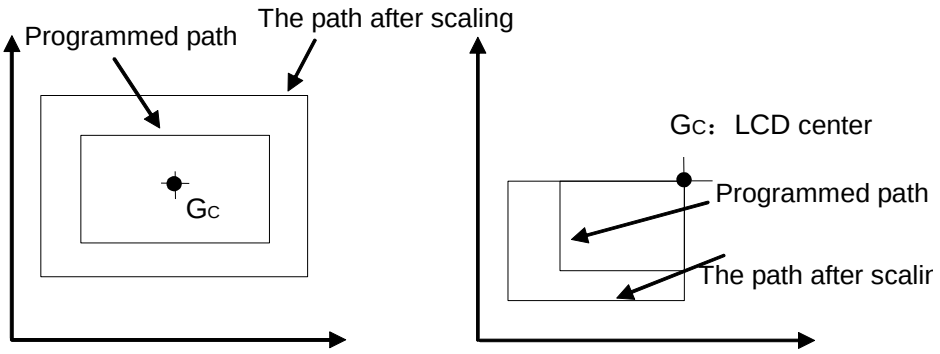
The maximum dimension of graph on the LCD screen is 205 mmX141 mm. If the workpiece shape is larger than the maximum dimension, the graph must be scaled. The scaling proportion is 0.01 to 100.00. The method for scaling in 2-D space is as follows:

- Scaling Proportion = MIN (horizontal proportion, vertical proportion);
- Horizontal proportion = α/machined length in horizontal direction
- Vertical proportion = β/machined length in vertical direction
- α : 150 mm
- β : 90 mm

The basic point of scaling is at the center of the LCD screen.

NOTE

Z axis is programmed via radius, X axis via radius or diameter, which is selected by the system's radius/diameter programming selection function.



The tool path on the LCD screen is described by the work coordinate values for the tool movement. The work coordinate values corresponding to the center of LCD are as follows:

- GcX = (X max + X min)÷2
- GcZ = (Z max + Z min)÷2

The maximum or the minimum value in X or Z is set by Graphic Parameter.

10.1 Setting Graphic Parameter

Graphic Parameter must be set as showed in Figure III-27 before start-up.

1 Press the key GRAPH, Graphic Parameter is displayed on the screen.

```
GRAHIC                                        O0001     N0063
                    GRAPHIC  PARAMETER

      AXIS SEL.   =              0   (XZ:0 ZX:1)
      SCALERATE   =              0   (Auto Cal.)
      X CENTER    =          0.000   (Auto Cal.)
      Z CENTER    =          0.000   (Auto Cal.)
      X  MAX.ABS  =          0.000
      Z  MAX.ABS  =          0.000
      X  MIN.ABS  =          0.000
      Z  MIN.ABS  =          0.000


NO.    004=
```

Figure III-27

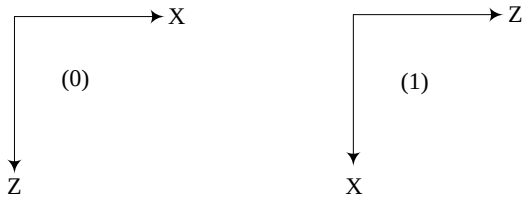
2 Press the cursor key ↓, ↑ and move the cursor under the parameter to be set.

3 Press the number key and the button INSRT. Input Graphic Parameter values.

4 (Repeat the procedure 2, 3) to set parameters to be set.

10.2 Meanings of Graphic Parameters

1 AXIS SEL (select coordinates system): Coordinate plane selection (XZ = 0, ZX = 1)



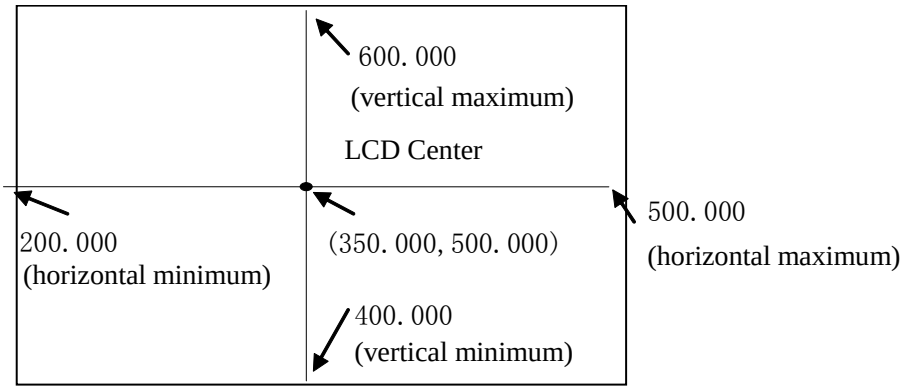
2 SCALERATE (scaling proportion): Set plotting proportion.

- Range: 1~10000
- 1 = 0.01 times

3 X/Z center (graph center): Set the coordinate values of LCD center in work coordinates system.

4 Maximum and minimum values: Set the maximum and minimum values of each axis, then corresponding graph center's coordinate values can be set automatically. Unit:0.001mm.

Example



Horizontal graph center =(500.000+ 200.000)÷2 = 350.000

Proportion= 150÷300 = 0.5

Vertical graph center =(600.000+ 400.000)÷2 = 500.000

Proportion= 90÷200 = 0.45

Scaling proportion = min{ 0.68,0.71 }= 0.68

It can be set in values no more than 68.

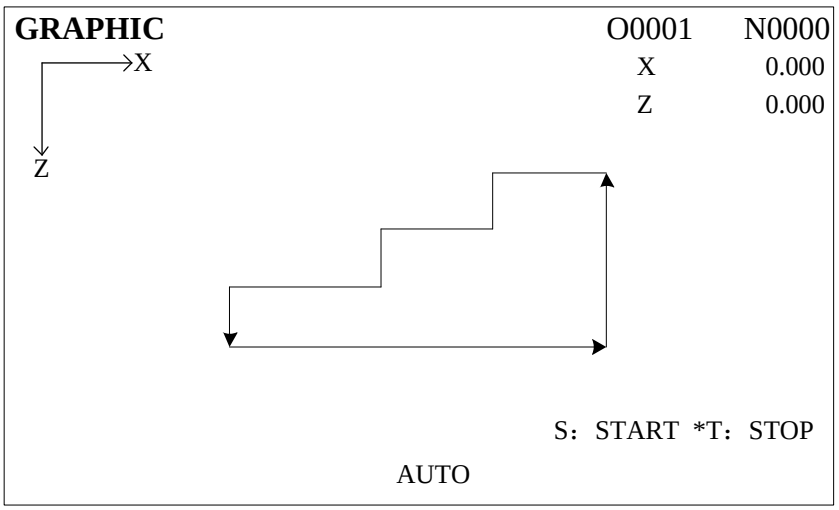
If it is necessary to modify graph center's parameter, it can be set after setting the maximum and minimum values of each axis.

10.3 Description of Tool Path

Drawing screen is on the second page of the Graph screen, it can be selected by the key 

or 

:



1 Press key S to enter Drawing mode, '*'move to S: START (Graphing).

2 Move machine tool in AUTO/MDI/JOG mode. When the absolute coordinate values change, the tool path will be displayed accordingly.

3 Press key T to enter graphing stop mode,'*'move to T: stop graphing.

4 Press key R to delete the graph that has been plotted.

NOTE

1 In program debugging, machine lock and dry run feed rate can be used.

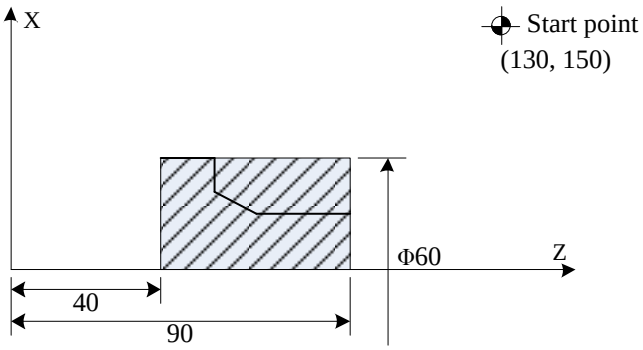
2 Top left corner on display is work coordinate value, the graph is drawn with this coordinate value.

10.4 Example

Diameter programming: (X axis)

Programmed unit: 0.001mm

Graphing plane 1



Plotting all tool paths:

Set the maximum and minimum values:

X max = 130000, X min = 0

Z max = 150000, Z min = 0

Graph center is set automatically.(65000, 75000)

Scaling proportion (horizontal) = 150÷150 = 1

Scaling proportion (vertical) = 90÷130 = 0.69

Scaling proportion coefficient is no more than 1.08 (108)

Only plotting the shadow

X max = 30000, X min = 0

Z max = 90000, Z min = 40000

Graph center is set automatically(15000, 65000)

Scaling proportion (horizontal)= 205÷50 = 4.1

Scaling proportion (vertical)= 141÷30 = 4.7

Scaling proportion coefficient is no more than 4.1 (410)

Offsetting the tool path as a whole on the LCD screen

Change the maximum and minimum with the same data: max+ a, min+ a

a < 0, plotting position offsets upward or rightward

a > 0, plotting position offsets downward or leftward

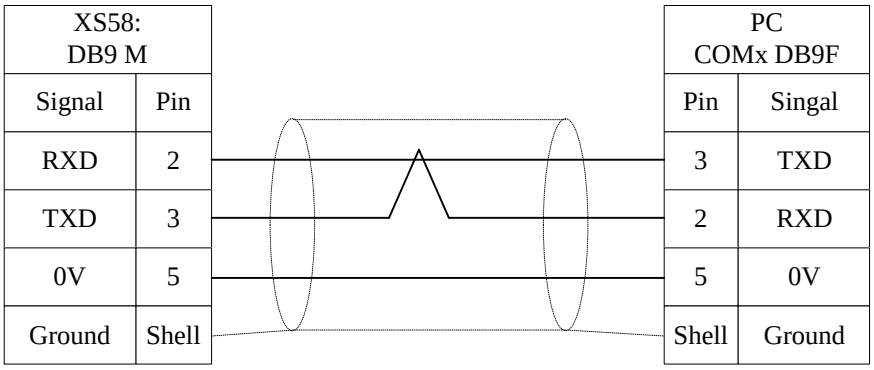
11 RS232 COMMUNICATION

11.1 Preparation

11.1.1 Connection of communication cable

The communication cable should be connected when CNC and PC are power OFF to avoid any hardware damage.

The communication cable diagram is shown in Figure III-28.



Line: RVVP 3 x 0.3 mm^2 (twisted pair line)

Figure III-28 Definition of RS232 cable

11.1.2 Parameters

042				PUCH					

5 PUCH Select the serial port communication function

0: Invalid

1: Valid

NOTE

It is necessary to reboot the system to activate the setting.

056	BAUTE: Baud rate of RS232 port

[Range] 2400, 4800

[Unit] bps

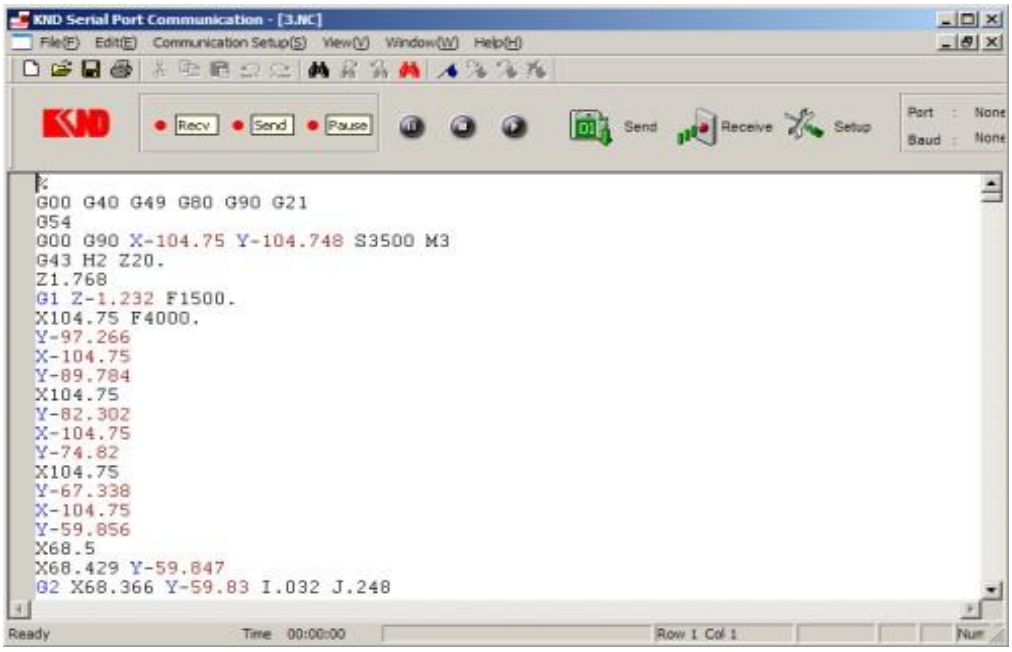
[Def. value] 2400

NOTE

It is able to transmit parameters and programs through RS232 port. Transmitting tool offset values is not supported.

11.1.3 KND serial port communication

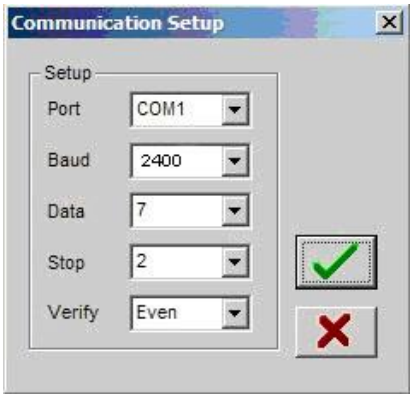
Software interface



Preparation

1 Click【file】→【new】 to create a file.

2 Click 【setup】to set COM port, the set interface is as that shown in below figure.



COM baud rate : 1200, 2400, 4800, 9600, 14400, 19200. Communication default setting: baud rate:2400, data bit :7, even, 2 stop bits.

11.2 Output Programs

11.2.1 Output one program

This is for transmitting a registered program in the memory to a computer. After connecting a PC,

1 In EDIT mode, press the PRGRM button to display program.

2 Run communication software in PC, select communication port, set same baud rate between CNC and PC, make PC be in 'waiting inputting' state.

3 Press the address key O, input the program number.

4 Press the key OUTPUT to transmit the corresponding program to PC.

NOTE

Press the key RESET to stop outputting.

11.2.2 Outputting all programs

It is to transmit all registered programs in the memory to a computer.

1 Set the mode selector switch to EDIT, press the PRGRM button to display program.

2 Run communication software in PC, select communication port, set same baud rate between CNC and the PC, make PC be in 'waiting' state.

3 Press the address key O, and press the keys –9999

4 Press the key OUTPUT to transmit all programs to PC.

11.3 Input Programs

11.3.1 Edit program on PC

1 To edit program files on PC, "Notebook" or "UltraEdit" editing software shall be used.

2 First line is :"%".

3 Second line is ": 1234", namely :＋program number; this line can be omitted.

4 Then input program line by line, and press EOB key to a new line.

5 Last line of program is: "%".

11.3.2 Input a program

1 Turn on the program protection switch.

2 Set the mode selector switch to EDIT, press the PRGRM button to display program.

3 Press the address key O and the program number.

4 Press the key INPUT, display "input"

5 Run communication software in PC, select communication port, set same baud rate between CNC and PC

6 Select the program and set the program to output status in the KND communication software. Then the program is transmitted from PC to CNC.

NOTE

If the program on PC contains program number and no change is required, step 3) can be omitted.

11.3.3 Input multiple programs

It is to input the contents of a file comprising several programs into the CNC system. The format of the file is as follows.

```
%
:1111;
……;
…….;
M30;
:2222;
……;
……;
M30;
:3333;
……;
……;
M30;
%
```

11.4 Comparison Programs

1 Turn off the program protect switch.

2 Set the mode selector switch to EDIT or AUTO, press the PRGRM button to display program.

3 Transfer the to-be-compared program in CNC.

4 Press the key INPUT, it displays “Compare”

5 Run communication software in PC, select communication port, set same baud rate between CNC and PC

6 Transfer the to-be-compared program in PC, make it in the state of outputting, and then begin comparing the two programs in PC and in CNC.

NOTE

Alarm P/S No.79 occurs and comparison stops when difference occurs.

11.5 Output Parameters

It is to transmit the registered program in the memory to a computer.

1 Set the mode selector switch to EDIT, press the PARAM button to display parameter.

2 Run the communication software, select an available serial port and specify the same baud rate with that of CNC. Click the [READ] button to wait for reading data from CNC.

3 Press key 

to transmit the parameter to PC.

11.6 Input Parameters

1 Turn on the parameter protect switch.

2 Set the mode selector switch to EDIT, press the 

button to display parameter.

3 Press the 

button.

4 Run communication software in PC, select communication port, set same baud rate between CNC and PC, make PC be in ‘waiting inputting’ state.

5 Select the parameter file and set it to output status. Then the parameter is transmitted from PC to CNC.

12 DISK OPERATION

It is possible to read and write U disk with the USB interface of the system. It is possible to output to /input from an U disk any parameters, tool compensations and programs.

When an U disk is inserted into the USB interface of the system, and the U disk is recognized by the system, the function of the serial port will become invalid. In this case, both inputting and outputting are done on the U disk. At inputting/outputting, it displays “U disk inputting" or “U disk outputting” at the lower right corner of the screen. When the prompt disappears, it means the inputting/outputting ends.

When no U disk is inserted into the USB interface of the system, or the U disk is not recognized by the system, the function of the serial port is valid. In this case, both inputting and outputting are done through the serial interface. At inputting/outputting, it displays “Serial port inputting" or “Serial port outputting” at the lower right corner of the screen. When the prompt disappears, it means the inputting/outputting ends.

The U Disk supports FAT files and FAT32 files.

12.1 Upgrade Software When Power-on

Enter the upgrade screen

Press 

and 

at the same time when turning on the power of the system. After entering into the Upgrade screen, pressing  and  cursor can adjust the brightness of the LCD.

Insert a U Disk

At the first line of the screen, it prompts the way for changing the prompting language. If it is not necessary to change the prompting language, one can just proceed without carrying out the said operation. The system will detect whether U disk is inserted, if no, it will keep on waiting. As that shown in Figure III-29:

Figure III-29

Upgrade software

When an U disk is detected, the system will detect whether it has a program in the current system. If yes, it will prompt the program version, otherwise it will display "No program or invalid version no.".

The program will detect and list the version number of files usable in the KND directory in the U disk.

For example: 1----- K100Ti-B_A02_120516

Any program other than upgrading program will not be listed. Now it will prompt to press the corresponding numerical key 1~8 to select the upgrading program. After pressing the corresponding numerical key, the upgrading starts. The upgrading is provided with a prompt in progress by percentage. If selecting numerical key '0' at this time, you can back up the current program of the system to the U disk. It is strongly suggested that a user will first backup the old version before upgrading to a new version.

During the upgrading, the following procedures are performed sequentially:

- **Step I**

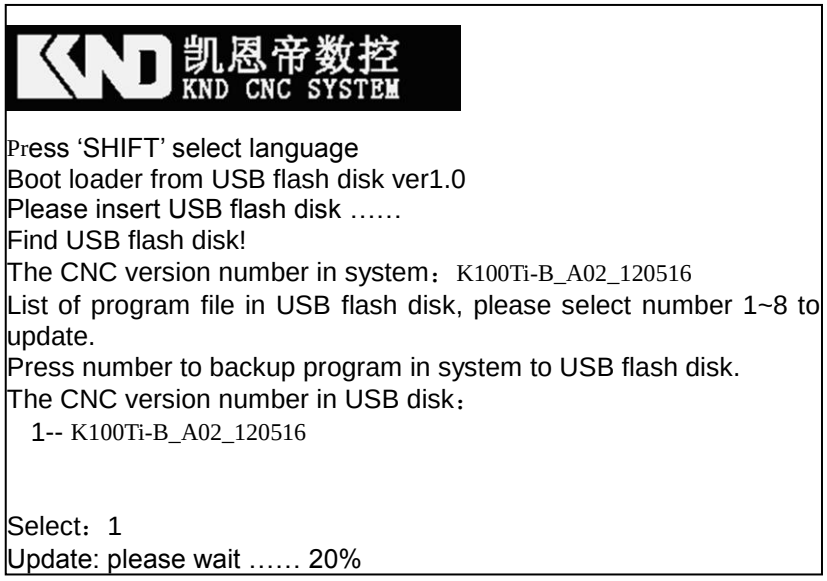
The files in the U disk are read in and checked. If the check fails, it will prompt "File test fails, please check the files"; after the checking succeeds, go to step II.

- **Step II**

The current program in the system is erased, then a new program is written. If the writing fails, it will prompt "the program writing fails, please retry or contact KND ", after the writing succeeds, go to step III.

- **Step III**

The program written a moment ago is checked again. If the checking fails, it will prompt "upgrading fails, please retry or contact KND"; if the upgrading succeeds, it will prompt "Upgrading succeeds, the system program will be automatically restarted after 3 seconds."



Additional descriptions

1 After inserting a U disk and the system prompts "An U disk is found", if you find it is not necessary to upgrade, you can press 

to let the program to go to system software (KND doesn't suggest this operation, instead KND suggests restarting; if the system software is confirmed to be right, this operation is also OK).

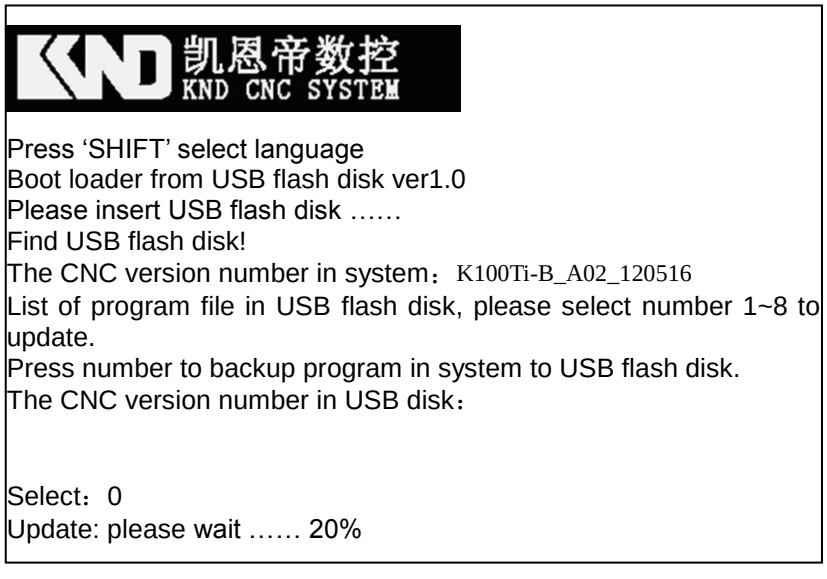
2 Program backing up

During the program upgrading, if you select the numerical key '0' to back up the program currently in the system to the U disk, it will display as follows:

After the backing up succeeds, it will prompt the user to select Program Upgrading, the subsequent operation is same as that for normal upgrading.

The file name saved at backing up takes a suffix 'bin'.

For example: 1——K100Ti-B_A02_120516



12.2 Operations

The following will introduce the inputting/outputting operation in respect to the U disk.

12.2.1 Transmit programs

Output programs to a U-disk

There are 3 ways for exporting a program(s) to the U disk:

- **Output a single program (same as former operation)**

1 In EDIT mode and Program screen, type in program number “Oxxxx”.

2 Press 

key, it prompts “U-disk Outputting” at the right lower corner of the screen.

3 When the prompt disappears, it means the outputting ends. In this case, file “Oxxxx.PRG” is generated under KND directory in the U disk.

- **Output a single program**

1 In EDIT mode and Program screen, select the program no. “Oxxxx” by the cursor.

2 Press 

key, it prompts “U-disk Outputting” at the right lower corner of the screen.

3 When the prompt disappears, it means the outputting ends. In this case, file “Oxxxx.PRG” is generated under KND directory in the U disk.

- **Output all programs**

1 In EDIT mode and Program screen, type in “O-9999”;

2 Press 

key, it prompts “U-disk Outputting” at the right lower corner of the screen.

3 When the prompt disappears, it means the outputting ends. In this case, each program in the system generates a file “Oxxxx.PRG” under the KND directory in the U disk, xxxx means program no.

Input programs from a U-disk

There are 4 ways for importing a program(s) from the U disk:

- **Importing a program from a file (same as former operation)**

1 In EDIT mode and Program screen, Program Switch ON;

2 Type in “Oxxxx”;

3 Press 

key, it prompts “U-disk Inputting” at the right lower corner of the screen.

4 When the prompt disappears, it means the inputting ends. In this case, the program in the file “Oxxxx.PRG” under the KND directory in the U disk is imported into the system, and the program no. is “Oxxxx”.

- **Importing a program from a file with the program no. in the file remained:**

1 In EDIT mode and U-Disk screen, Program Switch ON;

2 Move the cursor to select the file “Oxxxx.PRG” to be input;

3 Press 

key, it prompts “U-disk Inputting” at the right lower corner of the screen.

4 When the prompt disappears, it means the inputting ends. In this case, the program in the file

"Oxxxx.PRG" under the KND directory in the U disk is imported into the system, and the program no. maintains the program no. in the file "Oxxxx.PRG". If the program has no program no. in the file, then it is automatically allocated with no. O0001.

- **Importing a program(s) from a file:**

1 In EDIT mode and U-Disk screen, Program Switch ON;

2 Move the cursor to select the file "Oxxxx.PRG" to be input;

3 Type in "Oyyyy";

4 Press 

key, it prompts "U-disk Inputting" at the right lower corner of the screen.

5 When the prompt disappears, it means the inputting ends. In this case, the program in the file "Oxxxx.PRG" under the KND directory in the U disk is imported into the system, and the program no. is "Oyyyy".

- **Importing all programs from the U disk.**

1 In EDIT mode and U-Disk screen, Program switch ON;

2 In the Program Display screen, type in "O~9999";

3 Press 

key, it prompts "U-disk Inputting" at the right lower corner of the screen.

4 When the prompt disappears, it means the inputting ends. In this case, all the programs in all files in format "Oxxxx.PRG" under the KND directory in the U disk are imported into the system, and the program no.s are the program no.s in the files.

NOTE

1 During the operation, pressing 

can cancel the operation.

2 When importing a program from the U disk, the following alarms are same as that when importing from a serial port:

- Program no. already exists;
- CMOS space is full;
- Invalid program no.;

3 Turn off the program switch, it will execute the operations same as that for program inputting. The existing programs in the system will be compared with the programs in the file. The comparison is same as former one. During the comparison, the state displays "Programs comparing".

12.2.2 Transmit parameters

Output parameters

1 In EDIT mode, Parameter page;

2 Press 

key, it prompts "U-disk outputting" at the right lower corner of the screen.

3 When the prompt disappears, it means outputting ends. The file"K100Ti-B.PAR" is created in the KND directory in the U disk.

Input parameters

- **Method 1**

1 In EDIT mode and Parameter screen, Parameter Switch ON;

2 Press 

key, it prompts "U-disk Inputting" at the right lower corner of the screen.

3 When the prompt disappears, it means the inputting ends. In this case, the file "K100Ti-B.PAR" in the KND directory in the U disk is imported into the system.

- **Method 2**

1 In EDIT mode and U Disk screen, Parameter Switch ON;

2 Move the cursor to select the file "K100Ti-B.PAR" to be imported;

3 Press 

key, it prompts "U-disk Inputting" at the right lower corner of the screen.

4 When the prompt disappears, it means the inputting ends. In this case, the file "K100Ti-B.PAR" in the KND directory in the U disk is imported into the system.

NOTE

During the operation, pressing 

can cancel the operation.

12.2.3 Transmit tool offset values

Output tool offset values

1 In EDIT mode and Tool Compensation screen;

2 Press 

key, it prompts "U-disk Outputting" at the right lower corner of the screen.

3 When the prompt disappears, it means the outputting ends. In this case, a file "K100T.OFS" is generated under the KND directory in the U disk.

4 There are 2 ways for importing tool compensation from the U disk:

Input tool offset values

- **Method 1**

1 In EDIT mode and Tool Compensation screen;

2 Press 

key, it prompts "U-disk Inputting" at the right lower corner of the screen.

3 When the prompt disappears, it means the inputting ends. In this case, the file "K100T.OFS" in the KND directory in the U disk is imported into the system.

- **Method 2**

1 In EDIT mode and U Disk screen;

2 Move the cursor to select the file "K100Ti-B.OFS" to be input;

3 Press 

key, it prompts "U-disk Inputting" at the right lower corner of the screen.

4 When the prompt disappears, it means the inputting ends. In this case, the file "K100Ti-B.OFS" in the KND directory in the U disk is imported into the system.

NOTE

1 If the output file exists in the KND directory in the U disk, the existing file will be rewritten.

2 If the input file doesn't exist in the KND directory in the U disk, or there is erroneous content in the file, the system will alarm PS58: File doesn't exist or erroneous file content.

3 During the operation, pressing 

can cancel the operation.

13 SAFETY OPERATION

13.1 Over-travel Protection

In order to avoid damaging the machine tool due to over-travel of X axis or Z axis, machine tool must be provided with over-travel protection.

13.1.1 Hardware overtravel protection

Limit switches are installed at max. travels of X/Z in +/- direction, and wiring is done as shown in Figure III-30, the system parameter MOT of No.001 Bit6 must be set to 0. When overtravel occurs, the travel limit switch is enabled, the system stops movement and overtravel alarm is displayed. In this case, move the worktable in reverse direction can move away from the travel switch. Press 

to release the alarm.

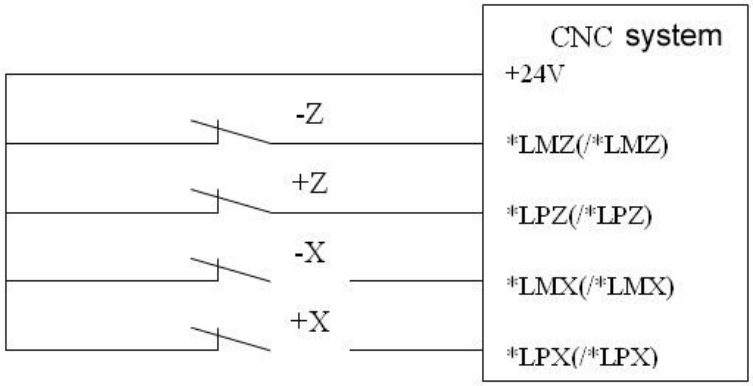


Figure III-30

13.1.2 Software overtravel protection

Software travel range is set via data parameter NO.015, NO.016, NO.017, and NO.018, and takes the machine tool coordinate value as reference, as shown in Figure III-31, X, Z are two axes of the machine coordinates system, NO.015, NO.017 are X axis +, - direction maximal travel, NO.016, NO.018 are Z axis +, - direction maximum travel. They are in the dashed frame is the range of software travel.

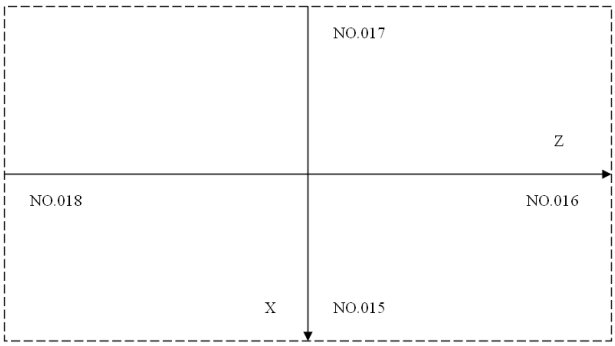


Figure III-31

If machine tool position (machine coordinate) exceeds the dashed area in Figure III-31, overtravel alarm will occur. Method for releasing overtravel alarm: move the axis in reverse direction (move in – direction for positive overtravel; and move in + direction for negative overtravel) and enter into dashed area, press 

, the alarm display will be released.

13.2 Emergency Operation

During machining, due to user program, operation and product defect, some accident may occur, then the system must be stopped immediately. This section explains CNC system management under emergency condition. As for emergency operation of CNC machine tool, refer to machine tool builder's instruction book.

13.2.1 Reset

When K100Ti-B outputs abnormally or axis acts abnormally, pressing 

, K100Ti-B will be in the reset state.

1 All axes stop moving;

2 M, S function outputting are disabled (through parameter setting, it is possible to set whether pressing 

can automatically turn off spindle CW/CCW, lubricating, and cooling signal);

3 Automatic run ends, modal function and state remain.

13.2.2 Emergency stop

Pressing EMERGENCY STOP (when external emergency stop signal is valid) when the machine tool is in a danger or under emergency status, CNC will enter into emergency stop state, and the machine tool stops immediately, all outputs (e.g. spindle and coolant) are off. Releasing emergency stop button and eliminating emergency stop alarm, CNC will enter into reset state.

NOTE

1 Ensue that malfunction have been remedied before releasing emergency stop alarm;

2 It can reduce electrical shock on the equipment by pressing emergency stop button before power ON and power OFF;

3 Return mechanical zero point again after releasing emergency stop alarm, so as to ensure correct coordinate position (if machine tool is not equipped with a mechanical zero, don't zero the machine);

4 Only the MESP of PA001 is set to 0, external emergency will be valid.

13.2.3 Feed hold

Pressing this key during operation can make machine tool pause. Note: this function cannot stop immediately the operation during thread cutting and cycle command running.

13.2.4 Cut off power supply

If any emergency or dangerous condition occurs when the machine tool is operating, you may immediately cut off machine power to avoid any accident. But the coordinates displayed in the CNC system may be different from actual position after power OFF, you must preset the tool again.

IV PART MACHINING

1 PART MACHINING

When using a CNC lathe to cut a workpiece, first of all, the location of the origin of the work coordinates system shall be defined, and the tool offset for each tool shall be established.

1.1 Setting Coordinate System

When machining a part on a CNC machine tool, the relative movement between the tool and the part must be within a certain coordinates system, only in this way can the machining with the specified programs be carried out. In order to ease the description on machine movement in programming, and simplify the programming, both the coordinates system and moving direction for the CNC machine tool have been standardized. According to JB3052-82 issued in 1982 by the Ministry of Machinery Industry from China, the two axes controllable by a CNC lathe is X and Z axis, which are vertical to each other and form a XZ plane Cartesian coordinates system as shown in Figure IV-1.

X axis: X-axis is defined as a direction vertical to the rotation central line of the spindle, and its positive direction is the direction that a tool goes away from the rotation center of the spindle.

Z axis: Z-axis is defined as a direction overlapped the rotation central line of the spindle, and its positive direction is the direction that a tool goes away from the headstock.

When a CNC lathe machines a workpiece, the coordinates system used are the machine coordinates system and the work coordinates system, which have the same coordinate axes and directions, and the only difference is the locations of their origins.

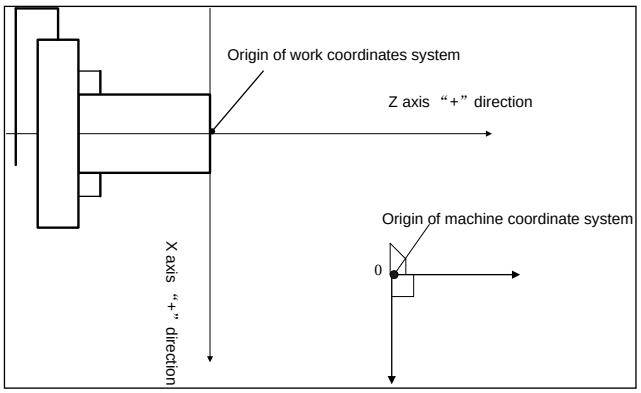


Figure IV-1 Machine coordinates system

1.2 Set Origin of Coordinate System

For K100Ti-B, there are two ways for setting the location of the origin of the machine coordinates system depending upon whether the machine has mechanical homing switches mounted or not.

Where mechanical homing switches are mounted, the location of the origin of the machine coordinates system is decided by the locations of the mechanical homing switches that are mounted at the points with the maximum travels at the positive directions of X and Z axis respectively. The locations of the mechanical homing switches are fixed. The location of the origin of the machine coordinates system is

also fixed. So long that the mechanical homing switches are not loose, each time at homing after power ON the machine, the tool will return to the same location.

Where no mechanical homing switches are mounted, a floating machine zero can be set (parameter ZRSZ/ZRSX is set as 0) by the following way: in JOG mode, moving the tool to a position that the tool will not collide the workpiece or other component at tool change and is properly located such that homing is easy, define it as the home of the machine, and set the machine coordinates at this point as 0. Way for setting: hold down the [CAN] key while pressing the address X and Z key, then the machine coordinates of X and Z axis are reset to 0, the point where the tool rests at this moment is set as the float mechanical zero of the machine.

NOTE

After setting the float zero, it will become effective only by confirming through mechanical homing. Under the Manual mode, first move the tool along the negative directions of the two axes away from the float zero set a moment ago, then carry out mechanical homing. In this way, the machine will return to the set float zero. For a CNC lathe without mechanical homing switch, after installing the CNC system, you first shall establish the floating mechanic zero. Normally it needs setting only once, and each homing will return to the same position.

1.3 Set Machine Coordinate System

K100Ti-B is provided with two tool-setting ways: namely, absolute tool setting, and relative tool setting.

1.3.1 Absolute tool setting

By absolute tool setting, it means the tool offset for each tool is independent. After returning to the machine zero manually, the location of the zero of the work coordinates system for the workpiece is automatically set, which is the mechanical zero of the machine. Now, if you program with absolute coordinates, all movements of the tool nose are relative to the zero of the machine coordinates system.

Generally, when machining a workpiece, the zero of the work coordinates system is set at the rotation center at the right end face of the workpiece. If you want to make the tip of each tool move relative to the origin of the work coordinates system, you have to set the tool compensation for each tool through tool setting, so as to offset the origin of the work coordinates system from the origin of the machine to the origin of the work coordinates system, which is normally the rotation center at the right end face of the workpiece.

As for the procedure for absolute tool setting, refer to Figure IV-2:

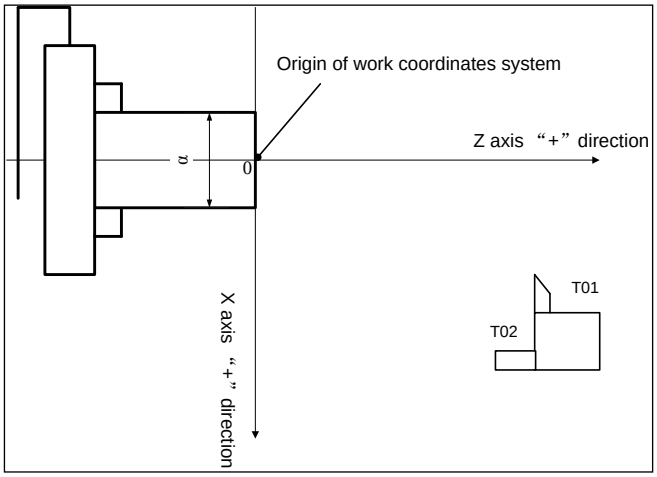


Figure IV-2 Part machining

1 Press  key to enter into "Mechanical homing" mode, and make the tool return to the home of the machine by carrying out mechanical homing for the two axes.

2 Select a tool, cut along the end of the workpiece in JOG mode. Move the tool to a safe location along the X axis with Z axis not moved. Stop the spindle.

3 Measure the distance from the end face of workpiece to the origin of the work coordinates system. If the origin of the work coordinates system is at left side of the right end face of the workpiece, you can directly input the positive value of the measurement into the tool offset no. in the【Measurement】page of the Tool Compensation, and the offset no. = offset no. of the value to be set + 100. If the origin of the work coordinates system is the center of the right end face of the workpiece, you can directly input Z0 into the tool offset no. in the 【Measurement】page of the Tool Compensation.

4 Cut along the outer diameter of the workpiece in JOG mode, move the tool along the Z axis to a safe position with X axis unmoved, stop the spindle, measure the diameter of the workpiece at the cutting point, and input the measurement into the corresponding tool offset no. in the 【Measurement】page of the Tool Compensation. If the cutting point is at back of the rotation central line of the workpiece, you shall input a negative measurement into the corresponding tool offset no.

5 Select manually another tool, repeat step (1) ~(4) to set the tool offsets for all tools.

Example for tool setting

Set the offset value into the offset unit with an offset no. 001, and make the origin of the work coordinates system be located at the center of the right end face of the workpiece. The workpiece diameter measured at the test-cutting point is 20 mm. After setting the tool, select the 【Measurement】page of the Tool Compensation, move the cursor to the offset no. of 101, input Z0, then press 

, input X20.0, press 

, now the tool offsets at X and Z directions are set.

In the system, it will automatically calculate the tool offsets in X and Z direction for each tool for offsetting the origin of the machine coordinates system to the origin of the work coordinates system in respect to each tool nose.

Example for machining

Machine a cylinder whose outer diameter is 20 mm and the length is 20 mm by two tools. The diameter of the blank is Φ25 mm. T01 tool is for cutting the outer diameter and acts as a standard tool; T02 is a parting tool with a width of 4 mm. (As that shown in Figure IV-2)

The program is as follows:

```
O0001；
N10 M03 S××；      Start the spindle
N20 T0101;         Change #1 tool
N30 G00 X20 Z5 ;   Positioning
N40 G01 Z-20 F100 ;
N50 G00 X50 Z50 ;  Move to tool-change position
N60 T0202;         Change #2 tool
N70 G00 X35 Z-24 ;Positioning
N80 G01 X-1 F80 ;  Cutting
N90 G00 X60 Z60 ;
N100 T0200;        Cancel offset of #2 tool
N110 M05；
N120 M30；
```

The operation procedures are:

1 Tool setting: first set the tool offsets for the two tools.

2 Running the program: you can start the program at any point. Take care that when automatic machining starts, the cursor in the program must be at the start of the program.

NOTE

1 When a program's first moving command is of absolute programming, and it has no G50, after finishing the setting of tool offset, retracting the tool to any point can start the program and carry out machining.

2 If the machine is provided with mechanical homing switches, every time when power ON the machine, you shall first return to the mechanical home, which can eliminate any accumulative error on the machine generated before. When any tool collision or emergency stop occurs hence the machine stops, you can restart the program for machining after returning manually to the mechanical home.

3 If the machine is provided with no mechanical homing switches, you can set the tools directly or start the programs for machining without mechanical homing after power on each time. When any tool collision or emergency stop occurs hence the machine stops during the machining, you have to set the tool and set the tool offset anew.

4 In order to prevent any malfunction when inputting the tool offsets, you can set OFMD2 of parameter P042 as 1. In this case, in the first page of the Tool Compensation screen, only address U/W can be input, and in the second page, only address X/Z can be input.

1.3.2 Relative tool setting

By relative tool setting, it is to take one tool nose as the datum point, the offset of which is 0, and the offsets for other tool noses will be set in relative to this reference tool nose.

When relative tool setting is employed, at the program start, operators must execute the command G50 Xα Zβ. Once the block specifying G50 is executed, the absolute commands following are based on the coordinate system established by G50 command. The work coordinates system set by G50 code has nothing to do with the machine coordinates system.

As for the procedures for the relative tool setting, refer to Figure IV-2:

1 Select a standard tool, which is generally the first tool to be used for machining.

2 Cut along the end of the workpiece in JOG mode. Move the tool to a safe location along the X axis with Z axis not moved. Stop the spindle.

3 Execute G50 Z0 in MDI mode and in Program screen (for which case, the origin of the work coordinates system is at the rotation center at right end face of the workpiece), then reset the W coordinate to 0 under the 【Relative Position】 screen.

4 Cut along the end of the workpiece in JOG mode. Move the tool to a safe location along the Z axis with X axis not moved. Stop the spindle.

5 Measure the workpiece diameter at the test cutting point, execute G50 Xx in MDI mode and in Program screen. (If the test cutting point is at the back side of the workpiece's rotation center, then execute G50 X-x in MDI mode).

6 Then reset U coordinate to 0 under 【Relative Position】 screen. Now the setting of the origin of the work coordinates system set by G50 command completes.

7 Change to another tool, in JOG mode, move the tool nose to the end face of the workpiece (that is, the test cutting point by the standard tool nose), move the cursor in the 【Measurement】 screen of the Tool Compensation to the corresponding tool offset no., input Z and then press the INSRT key, in this way, the setting of the relative tool offset in Z direction for the tool completes.

8 In JOG mode, move the tool nose to the outer diameter, i.e. the test cutting point of the standard tool, input X and then press the INSRT key, in this way, the setting of the relative tool offset in X direction for the tool also completes.

9 If other tools are to be set, you can repeat above steps by setting the tools and establishing the tool offsets in relative to the standard tool in X and Z direction.

After the tool setting, move the tool to the program start, i.e. the value of α β in G50 Xα Zβ; and （α, β） are the distances in X and Z axis of the standard tool nose to the origin of the work coordinates system. When the program ends, the written program must cause the standard tool to return to the program start for re-starting the program.

After the machine runs the program, the system will memorize the coordinates of the machine at the start point. If during the machining, the running aborts, you can return to the program start by way of Program Homing to continue the machining (as for the way of Program Homing, please refer to the section 3.8.1).

Example for machining

Also to machine the workpiece shown in Figure IV-2 by two tools. The program is as follows:

O0001；

N10 G50 X50 Z50；	Establish the workpiece coordinate system and define the start point
N20 T0101;	Change #1 tool
N30 M03 S××；	Start spindle

N40 G00 X20 Z5；	Positioning to (20, 5) in workpiece coordinate system
N50 G01 Z-20 F100；	Cutting
N60 G00 X50 Z50；	Positioning to tool-change position
N70 T0202；	Change #2 tool
N80 G00 X35 Z-24；	
N90 G01 X-1 F80；	
N100 T0100；	Change #1 tool and cancel tool offset
N110 G00 X50 Z50；	Return to the start point
N120 M05;	Stop spindle
N130 M30；	Program end

The operation procedures are:

Tool setting:

Define the T01 tool as the standard tool in the above-mentioned way, set the origin of the coordinates system at the right end face of the workpiece, and define temporarily the tool offset of the standard tool as 0. And set the tool offset of the T02 tool relative to T01 tool. If there is any dimensional deviation after machining, you can also adjust the tool offset for the T01, which will have no impact on the tool offset of T02.

Running the program:

After setting the tools, select the standard tool in JOG mode, execute the block G00 X50 Z50; in MDI mode, position the standard tool rapidly to a point that is X50 Z50 to the origin of the work coordinates system at right end face of the workpiece. Now you can run the program. At automatic running, the cursor must be located at the program start.

NOTE

1 When executing the block G50 Xα Zβ, the block will not cause the tool to move but just define the current position of the tool in the new work coordinates system, also define the position of the origin of the work coordinates system.

2 For the work coordinates system set by the G50 code, the tool-starting point for running the program every time must be the same point, that is, the position (α, β) in the work coordinates system set by the G50 code. If power is off before the tool returns to the tool-starting point, before running the program after restarting the machine, you have to set the tool and get the location of the tool-starting point for the standard tool. Way for determining the location of the tool-starting point for the standard tool: In MDI mode and Program screen, input the tool no. of the standard tool, e.g. T0100 to select the standard tool, and cancel its tool offset. Move the tool to the test cutting point in Manual mode, i.e. the right end face of the job for Z axis, and the outer diameter for X axis; execute G50 Xx Z0 in MDI mode (x is the job diameter at the test cutting point), now the location of the origin of the work coordinates system is defined; then execute G00 Xα Zβ in MDI mode and position the tool rapidly to the tool-starting point; now you can restart the program for machining the job.

1.3.3 Adjusting tool offsets

X axis: when the machined workpiece size (diameter) is bigger than the desired size, enter a negative

increment for tool offset (U-) into the corresponding tool offset no. in the 【Tool Compensation】 screen; when the machined workpiece size (diameter) is smaller than the desired size, enter a positive increment for the tool offset (U+). For example, if the actual measurement of the outer diameter machined by T01 is Φ20.02, and is 0.02 mm higher than the desired value, you can directly enter U-0.02 into the no.001 tool offset in the 【Tool Compensation】 screen to reduce the tool offset. On the next execution of the same program, system will cut more material

Z axis: for facing, when the machined workpiece size is longer than the desired size, enter a negative increment for tool offset (W-) into the corresponding tool offset no. in the 【Tool Compensation】 screen, so that the work coordinates system relative to the tool will offset leftward; when the machined workpiece size is shorter than the desired size, enter a positive increment for tool offset (W+), so that the work coordinates system relative to the tool will offset rightward. For parting, the way is opposite to above, for example, for the T02 parting tool mentioned above, after parting, the actual measurement of the workpiece is 20.1, which is 0.1 mm higher than the desired one, so W+0.1 shall be entered into no.002 in the 【Tool Compensation】 screen to make the workpiece shorter after parting.

V CONNECTION

1 STRUCTURE & INSTALLATION

1.1 Composition

K100Ti-B CNC system consists of following parts (as shown in Figure V-1)

1 CNC control unit

2 Motor driver

3 Motor

4 Insulating transformer

5 Electric cabinet

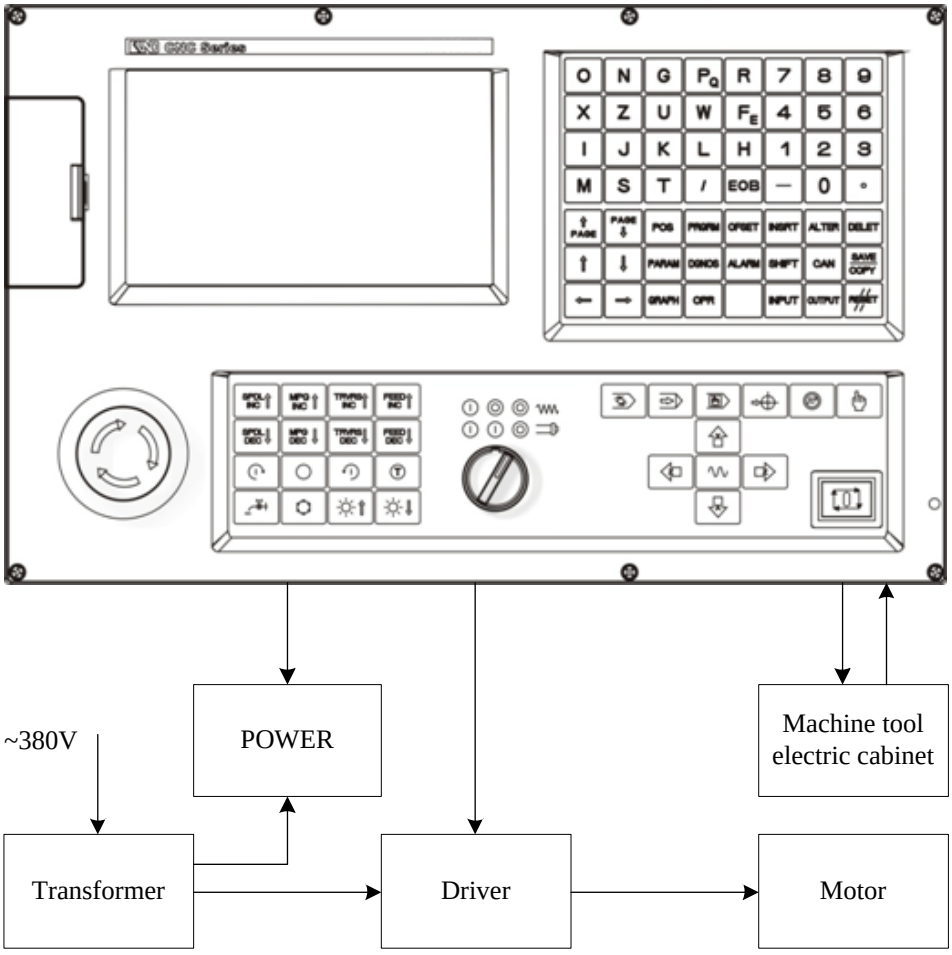


Figure V-1

1.2 System Panel

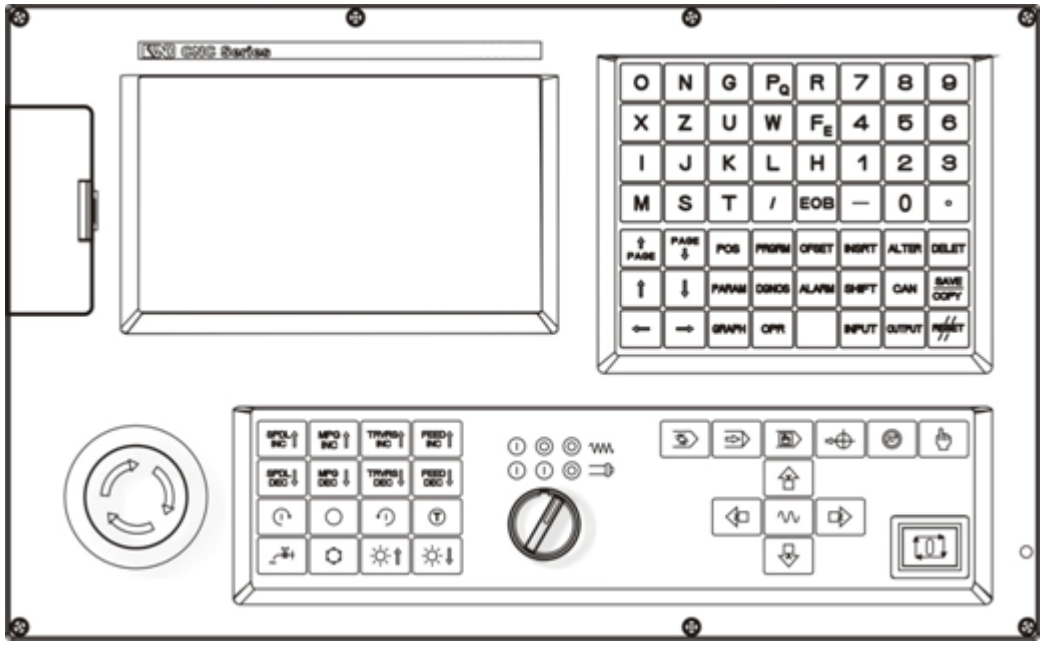


Figure V-2

1.3 System Installation Dimension

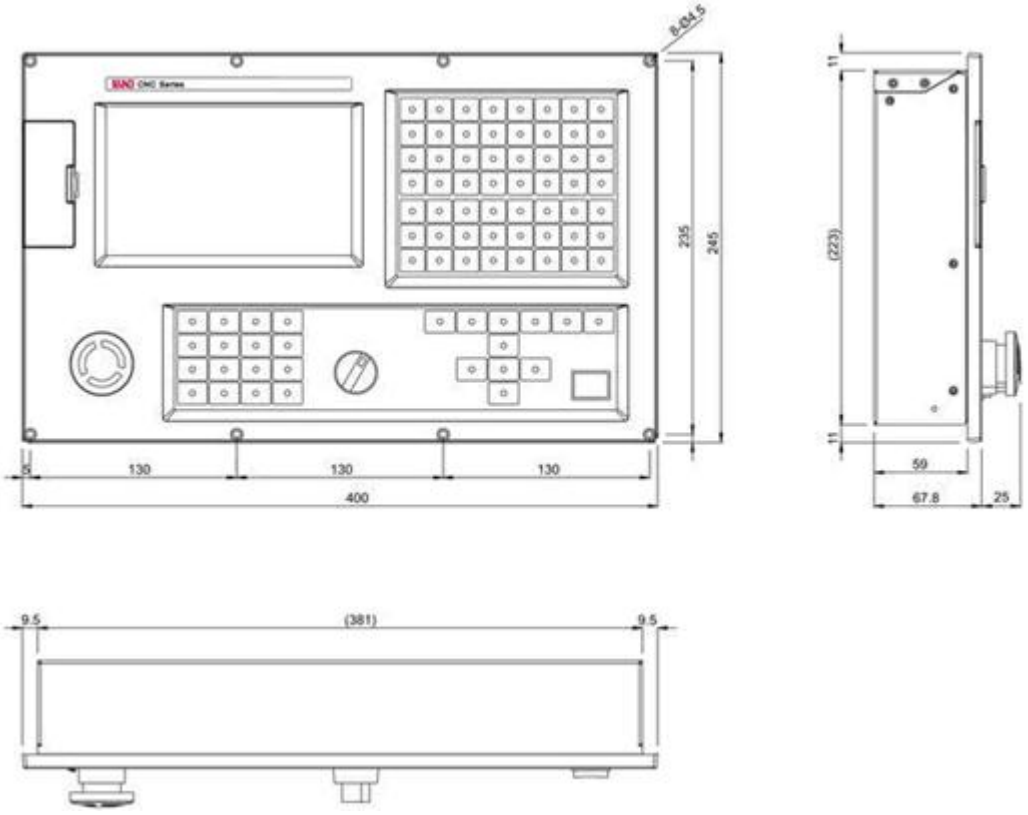


Figure V-3

1.4 Additional Operator's Panel Dimension

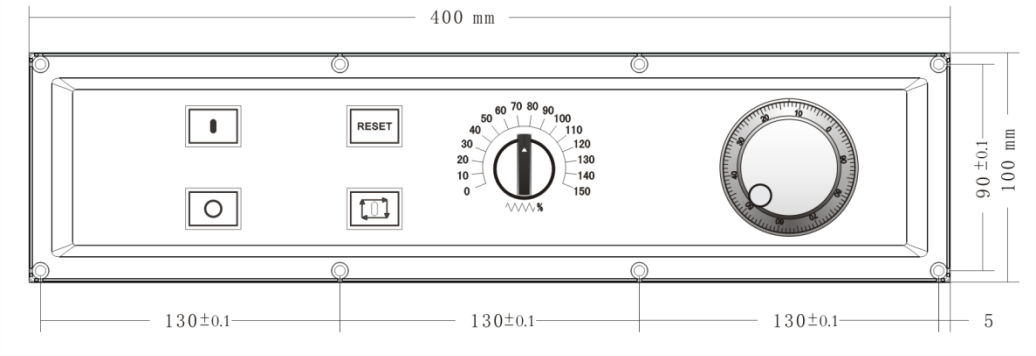


Figure V-4

1.5 Power Source Dimension

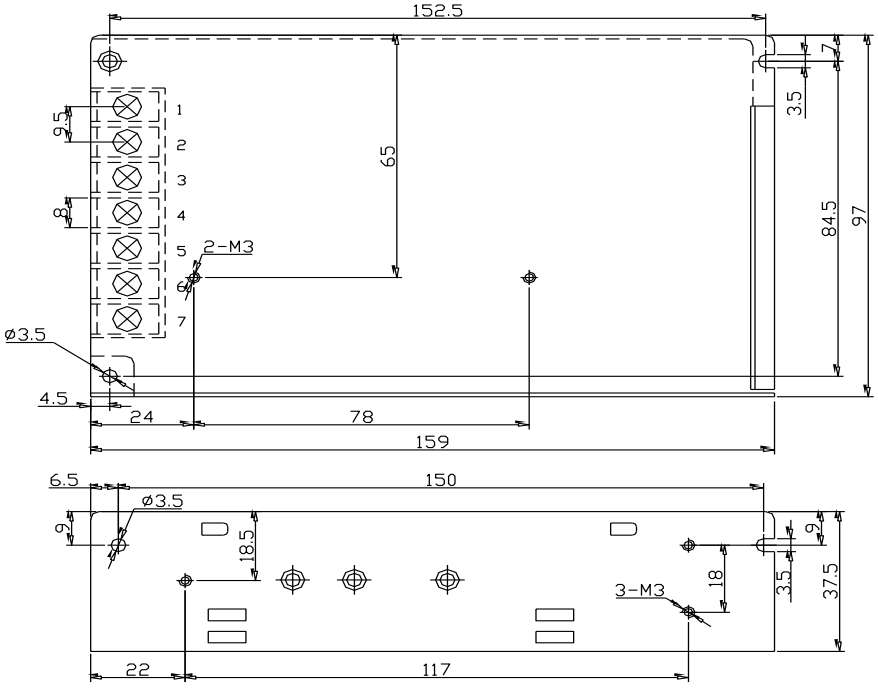


Figure V-5

1.6 Installation Conditions

1 The electric cabinet must be able to effectively prevent the entrance of any dust, coolant and organic solution.

2 The electric cabinet shall be so designed that the distance between the CNC rear cover and the cabinet box is not less than 20 cm, and the temperature difference between the interior and the exterior shall be ensured not more than 10 ℃ when the temperature rises in the electric cabinet.

3 In order to guarantee air circulation in the cabinet, a fan can be mounted in the cabinet.

4 The display panel must be mounted at a place that the coolant can't splash on.

5 The electric cabinet should be designed that any disturbance from the external air should be minimized and the transmission of any interference toward the CNC system should be avoided.

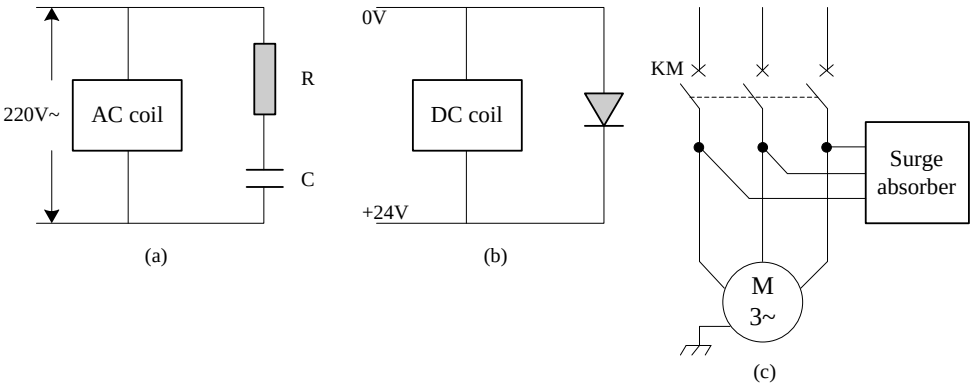
1.7 Method for Preventing Interference

At the design of the CNC system, such interference-immunizing measures as shielding electromagnetic radiation from the space, absorbing the shock current and filtering clutter in the power supply, etc. have been taken, which can avoid to certain extent the impact from any external interference source on the CNC system itself. In order to ensure that the CNC system can work steadily, the following measures are necessary when installing and connecting it:

1 The CNC system shall be far away from any equipment generating any interference (e.g. transducer, AC contactor, static generator, high-voltage generator and the sectioning device of a power line, etc.).

2 The CNC system shall be supplied with power through an isolating transformer. The machine tool on which the CNC system is installed must be grounded; the CNC system and the driver must be connected with an independent grounding line at the grounding point.

3 Suppressing the interference: a RC circuit shall be connected paralleled to the AC coil; the RC circuit shall be installed as near as possible to the inductive load; at the two ends of the DC coil, a follow-current diode shall be connected respectively in parallel way; at the winding end of the AC motor, a surge absorber shall be connected.



4 The lead-out cable of the CNC system shall employ twisted shielded cable or shielded cable; the shielding layer of the cable at the CNC side shall be grounded with a single end; and the signal line shall be as short as possible.

5 In order to reduce the interference between the CNC signal cables and that between the signal cables and the strong-current cables, the following principles shall be followed when laying the lines:

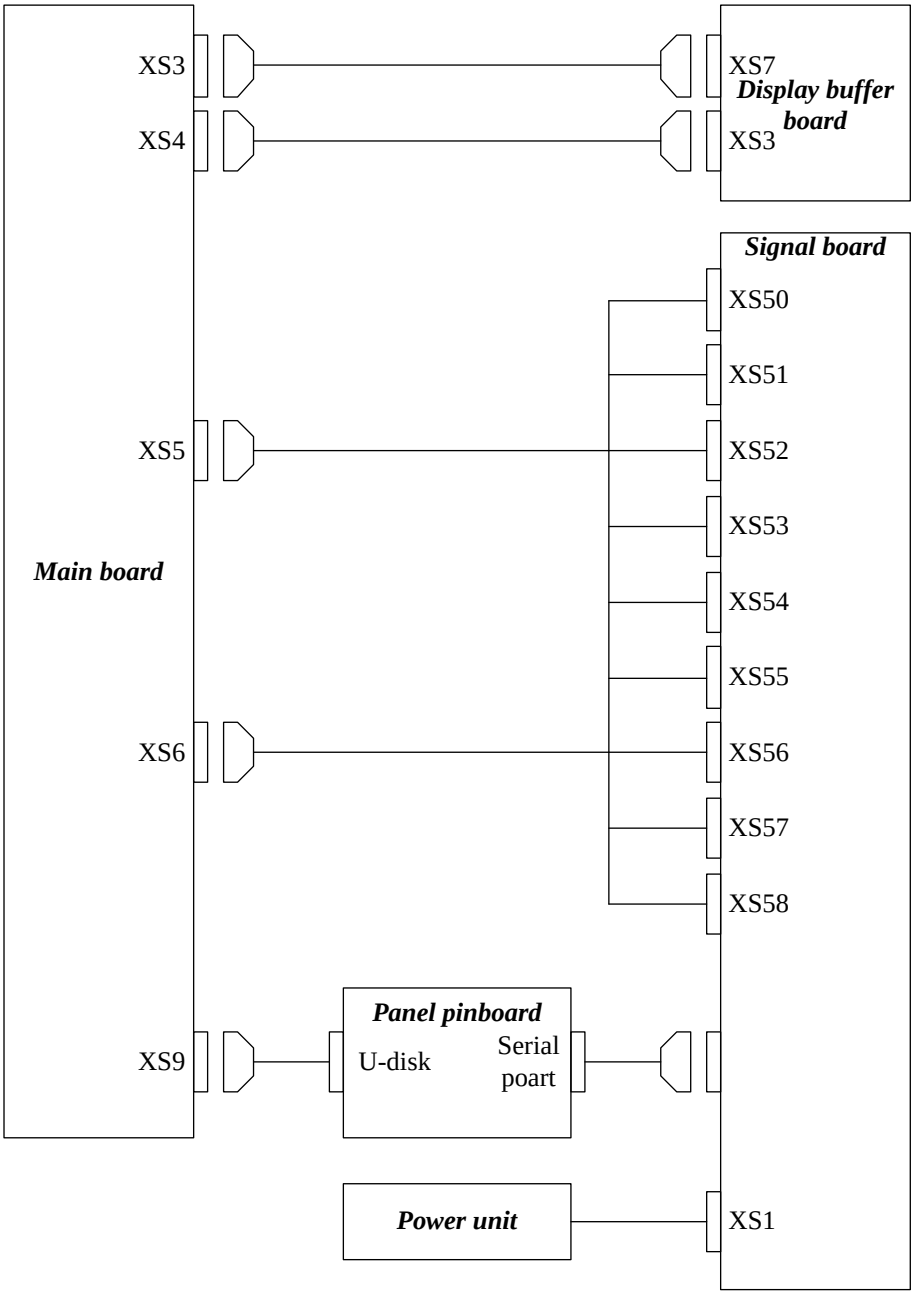
Group	Type of cable	Requirements on laying lines
A	AC power cord	Bind the cables in group A separated from the cables in group B and C, maintain the distance between them at least 10 cm, or carry out electromagnetic shielding for the cables in group A.
	AC coil	
	AC contactor	
B	DC coil (24 VDC)	Bind the cables in group B separated from the cables in group A or carry out electromagnetic shielding for the cables in group B; the cables in group B shall be far as
	DC relay (24 VDC)	
	Cable between the CNC system	

	and the strong-current cabinet	much as possible from the cables in group C.
	Cable between the CNC system and the machine	
C	Cable between the CNC system and the servo driver	Bind the cables in group C separated from the cables in group A or carry out electromagnetic shielding for the cables in group C; the distance between the cables in group B and the cables in group C shall be at least 10 cm; the cables are twisted lines.
	Position feedback cable	
	Position encoder cable	
	Handwheel cable	
	Other shielding cables	

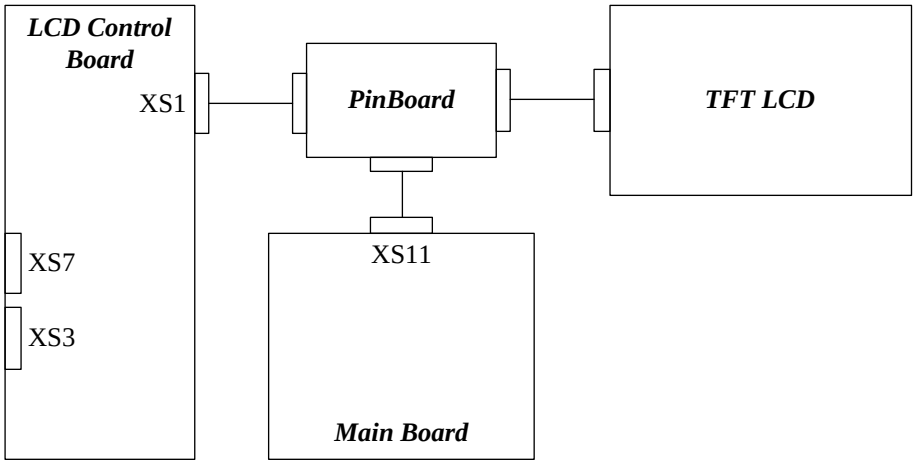
2 INTERNAL CONNECTIONS

2.1 Block Diagram

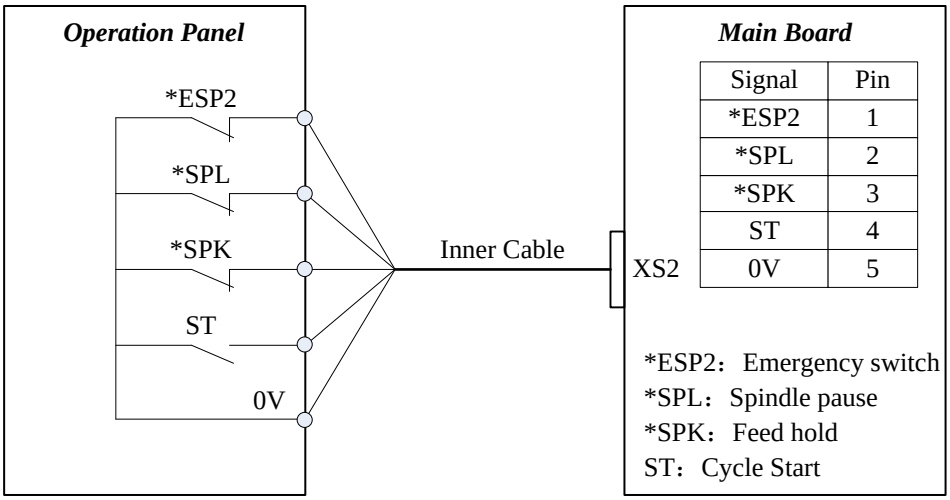
2.1.1 Main board connection



2.1.2 LCD control board connection



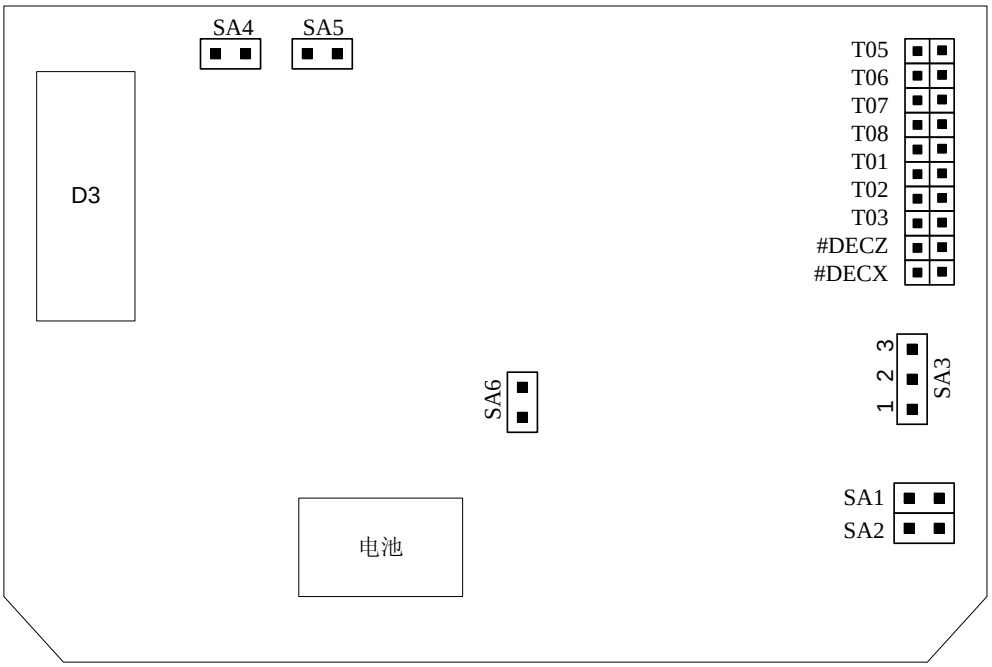
2.2 Operation Panel Switches Connection



2.3 Description of Switches On Main Board

2.3.1 Diagram of the main board

Main board version: 0020I-0200-W01Z-0704



2.3.2 Description of the switches

Switch no.	Switch state		Meaning	Remarks
SA1		Short circuited	The level of the one-rotation signal for Z axis to home is +5 V	The encoder provides the one-rotation signal For servo drive
		Open circuited	The level of the one-rotation signal for X axis to home is +24 V	The switch provides the one-rotation signal For step drive
SA2		Short circuited	The level of the one-rotation signal for Z axis to home is +5 V	The encoder provides the one-rotation signal For servo drive
		Open circuited	The level of the one-rotation signal for X axis to home is +24 V	The switch provides the one-rotation signal For step drive
SA3		1-2 short circuited	VP voltage is +24 V	For servo drive
		2-3 short circuited	VP voltage is +5 V	For step drive
SA4		Short circuited	Movement command output is in form of double pulses	The setting of the system should be consistent with that of

Switch no.	Switch state		Meaning	Remarks
		Open circuited	Movement command output is in form of pulse + direction	the drive
SA5/6		Open circuited	Fixed setting	
T1~T8 #DECX #DECZ		Short circuited	Inside the system corresponding to the input points, pull-up resistance is provided.	Refer to section 5.4.6

NOTE

1 Above-mentioned switches have been set by KND or the machine builder based on the configurations of the machine.

2 When a user configures the switches, please follow the above table.

3 VP voltage means the voltage supplied by the system to the driver.

The pin 12, 13 of socket XS51 (X axis) and XS52 (Z axis) in the rear cover board are the outputting end of VP voltage.

It depends upon the requirements of driver that VP can be set as +24V or be set as +5V.

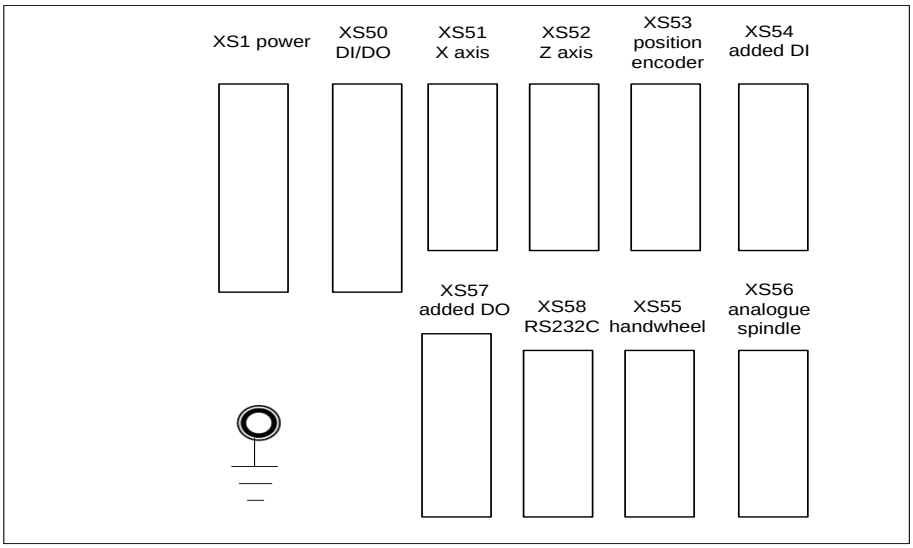
4 VP=+5V for the KND-BD3H-A/BD3D/BD3L/BD3S step driver. VP=+24V for AC servo driver, e.g. KND-SD100/SD200.

5 The pulse-receiving way of KND drivers is movement command pulse and movement direction signal.

6 Diagram of main board setting switch position.

3 EXTERNAL CONNECTIONS

3.1 Diagram of Sockets on Rear Cover Board



Description on interfaces:

XS1 : 5-pin green socket, power supply interface

XS50: DB25 female socket, I/O interface.

XS51: DB15 female socket, connected with X-axis driver.

XS52: DB15 female socket, connected with Z-axis driver.

XS53: DB15 female socket, connected with the spindle encoder.

XS54: DB15 male socket, additional inputting interface.

XS55: DB9 female socket, hand wheel interface.

XS56: DB9 male socket, analog spindle interface, connected with the transducer.

XS57: DB15 male socket, additional outputting interface.

XS58: DB9 male socket, RS232 communication interface;

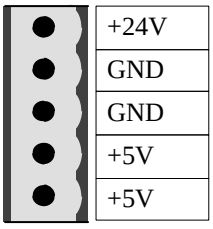
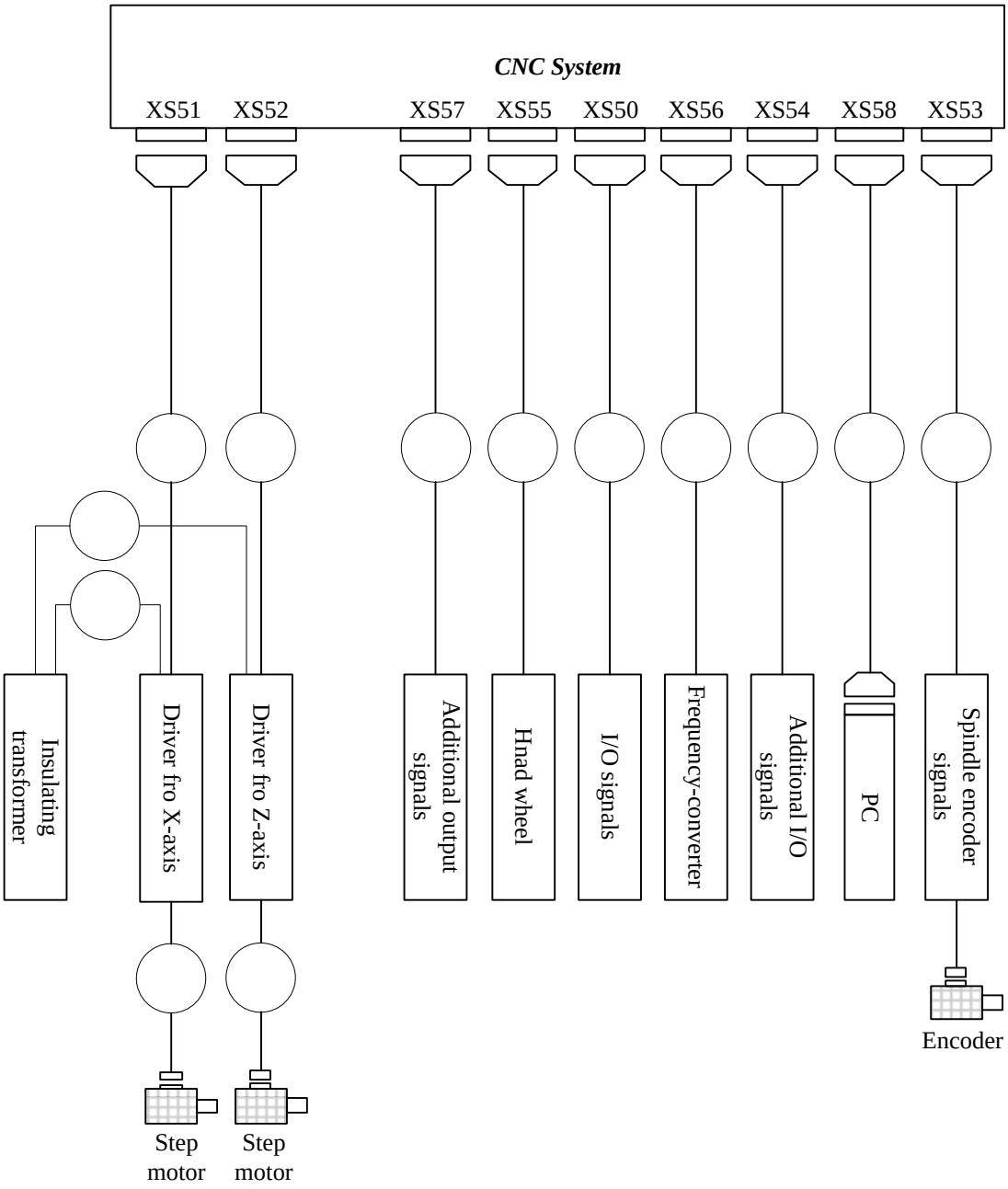


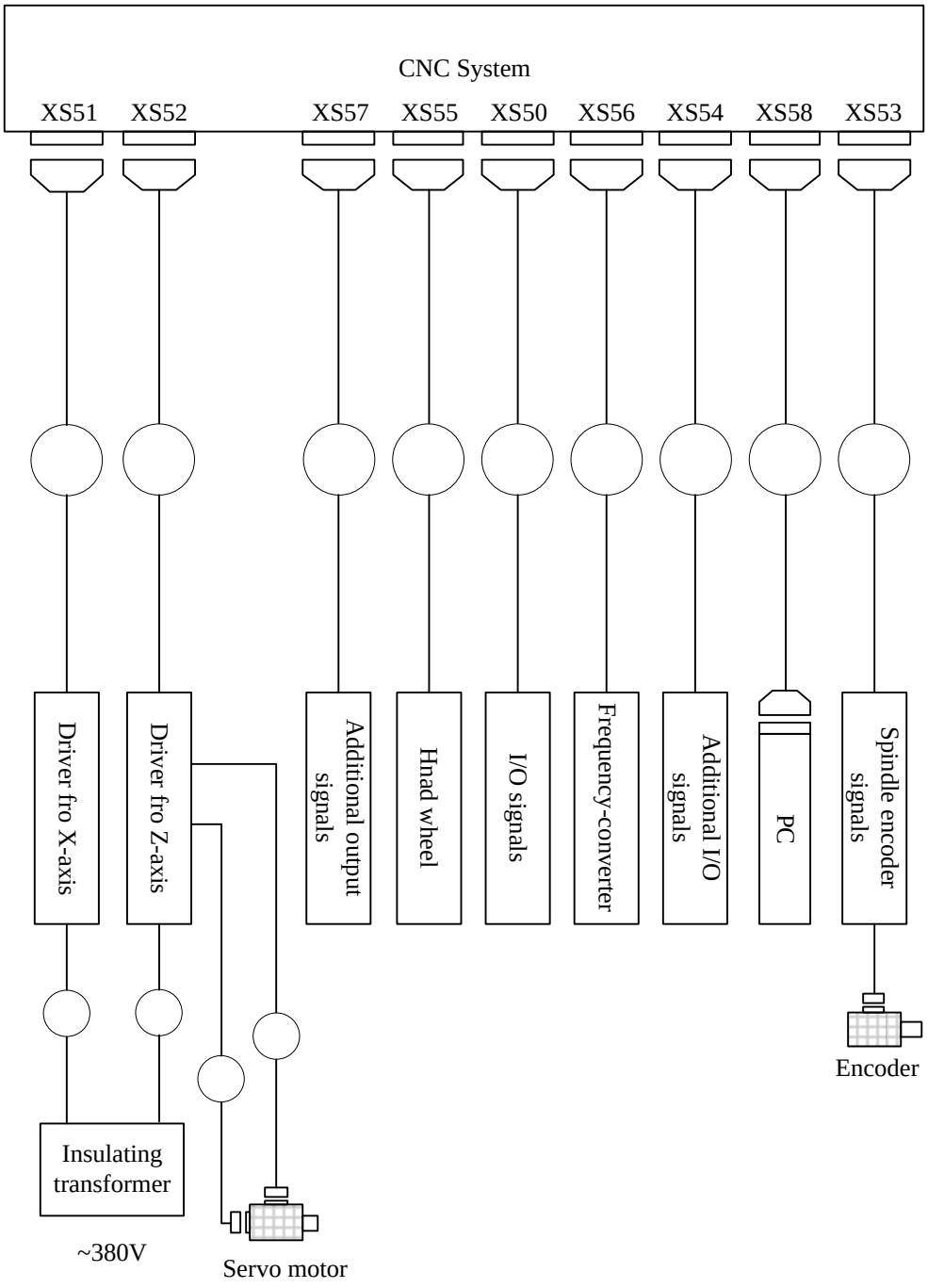
Figure V-6 Pin definitions of the power socket

3.2 Block Diagrams of External Connection

3.2.1 Equipped with Step Motors



3.2.2 Equipped with servo motors

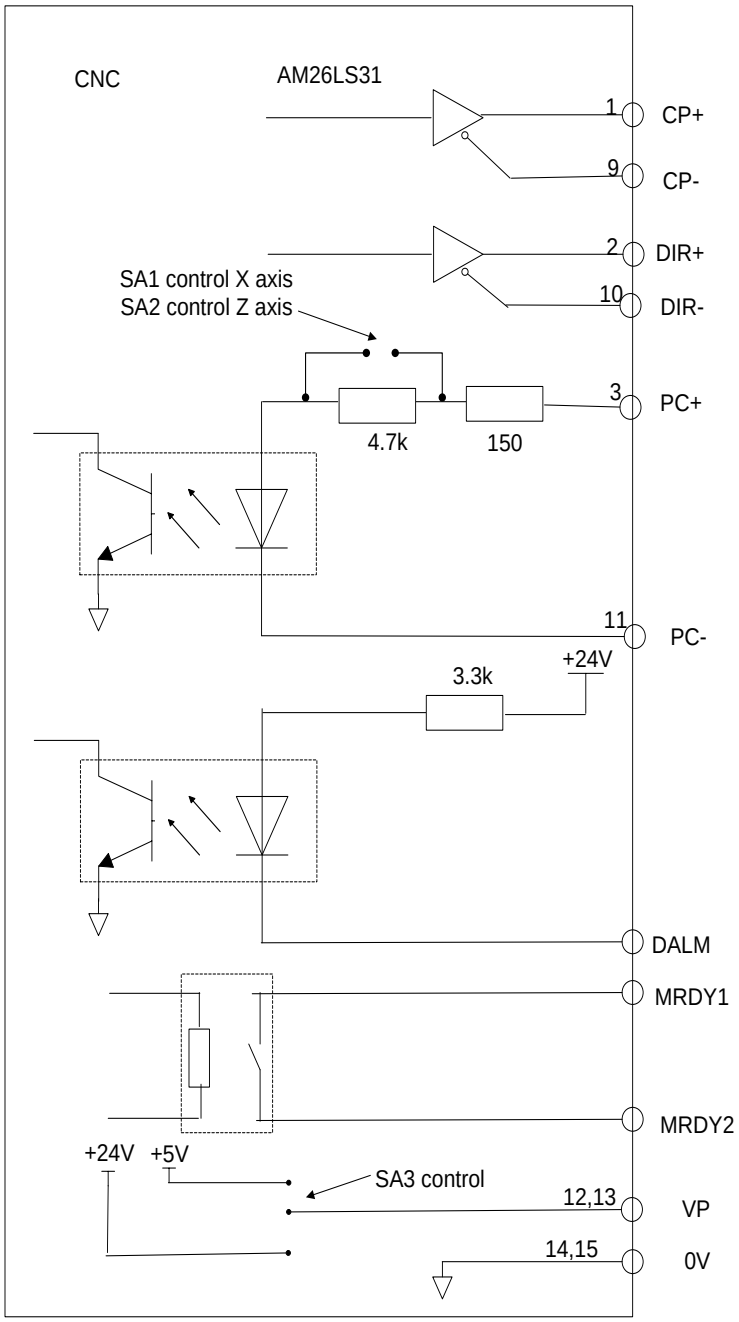


NOTE

Connections between servo driver for X-axis and servo motor are the same as those for Z-axis.

3.3 Connections from CNC System to Drivers

3.3.1 Block diagram



3.3.2 Signal list

Socket model at the system side is: DB15F (DB type 15-pin female)

Pin	Signal	Pin	Signal
1	XCP+	9	XCP-
2	XDIR+	10	XDIR-
3	XPC+	11	XPC-
4		12	VP
5	XDALM	13	VP
6		14	0V
7	XMRDY1	15	0V
8	XMRDY2		

Pin	Signal	Pin	Signal
1	ZCP+	9	ZCP-
2	ZDIR+	10	ZDIR-
3	ZPC+	11	ZPC-
4		12	VP
5	ZDALM	13	VP
6		14	0V
7	ZMRDY1	15	0V
8	ZMRDY2		

3.3.3 Descriptions (n represents axis name: X/Z)

3.3.3.1 Movement command signal

1 Single pulse output (SA4 open circuited)

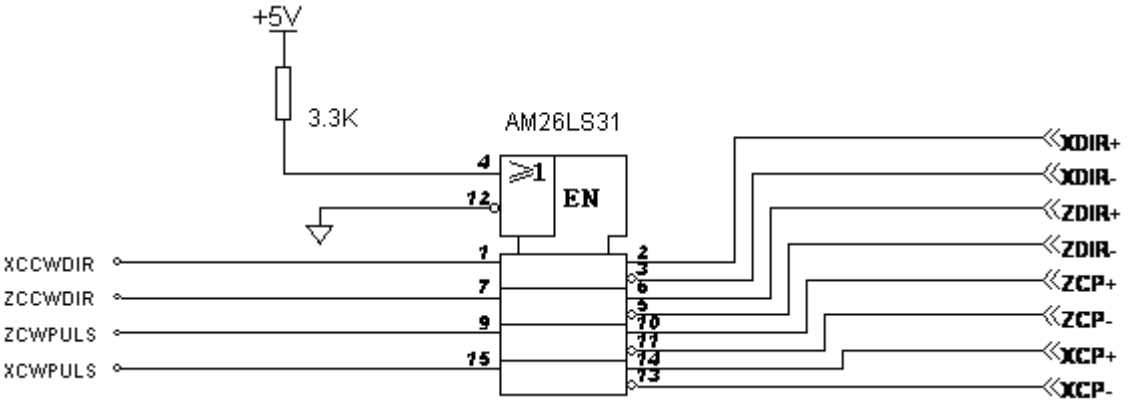
- nCP+, nCP-; nCP is the commanded pulse signal
- nDIR+, nDIR-. nDIR is movement direction signal.

Both the signals are of differential outputs.

2 Double pulse output (SA4 short circuited)

nCP in the signal list is a negative command pulse signal CCW, nDIR is a positive command pulse signal CW.

3 Diagram for the interfaces of the movement command signals

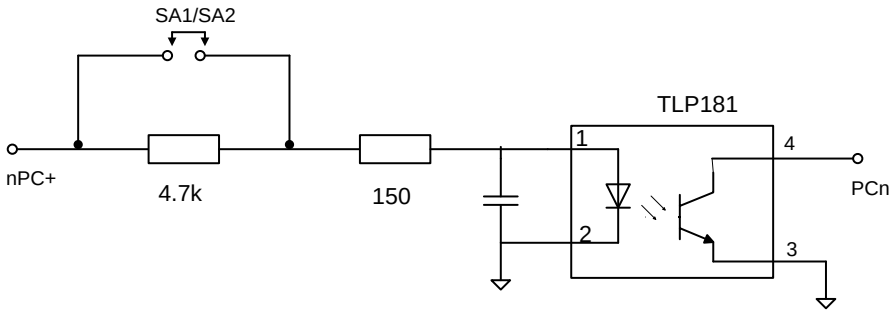


3.3.3.2 nPC+

The receiving circuit at the system side of the signal is as shown in the following:

With SA1/SA2 short circuited, the level of the one-rotation signal for homing (PC) is +5 V.

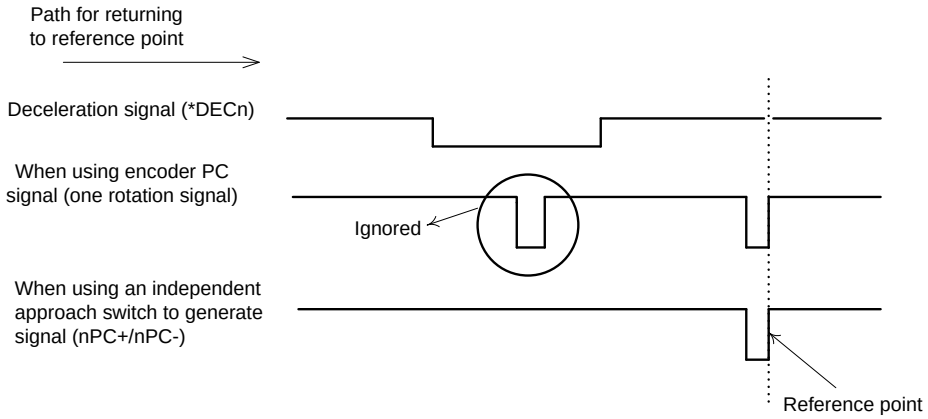
With SA1/SA2 open circuited, the level of the one-rotation signal for homing (PC) is +24 V.



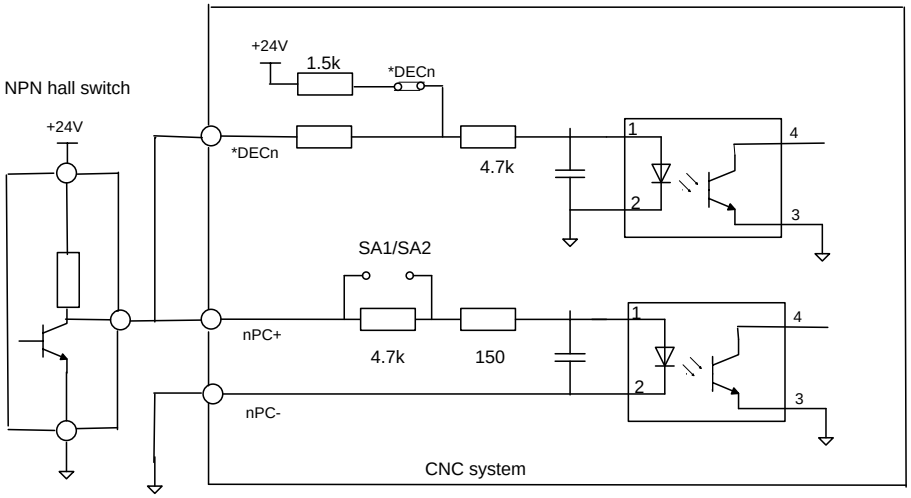
Homing way B

A user shall provide the homing deceleration signal *DECn and homing one-rotation signal nPC+.

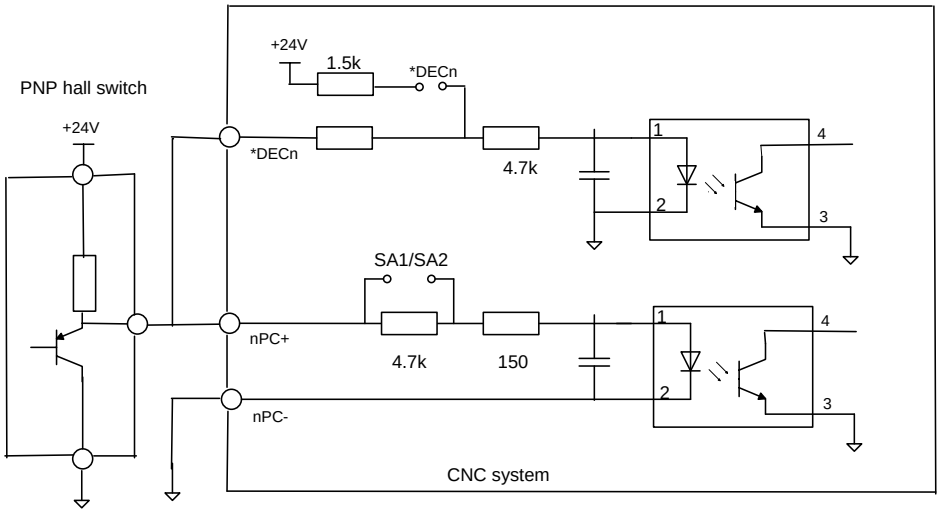
The waveform of the nPC+/ nPC- signal that shall be provided by the user is as follows:



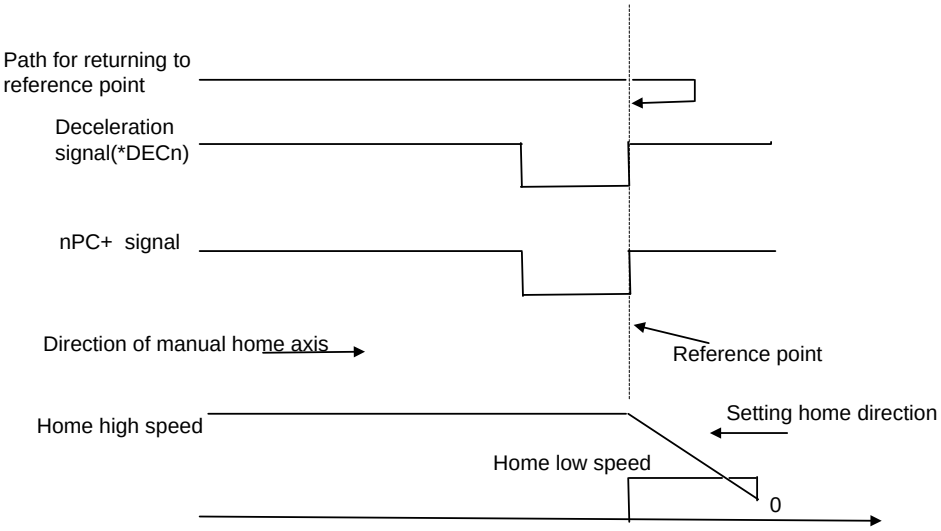
The connection way when a single NPN Hall proximity switch is used as both a deceleration switch and the home signal for the machine reference point is as follows:



The connection way when a single PNP Hall proximity switch is used as both a deceleration switch and the home signal for the machines reference point is as follows:



The homing process is as shown in the figure below:

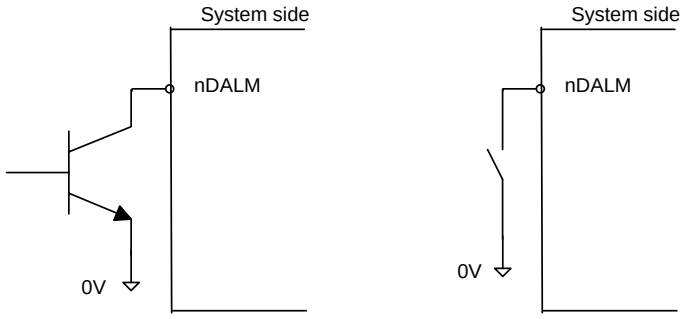


NOTE

If a Hall switch is used as the homing signal, the way is homing way C. The parameter ZRSX/Z and ZCX/Z shall be set as 1. Refer to the Parameter Description.

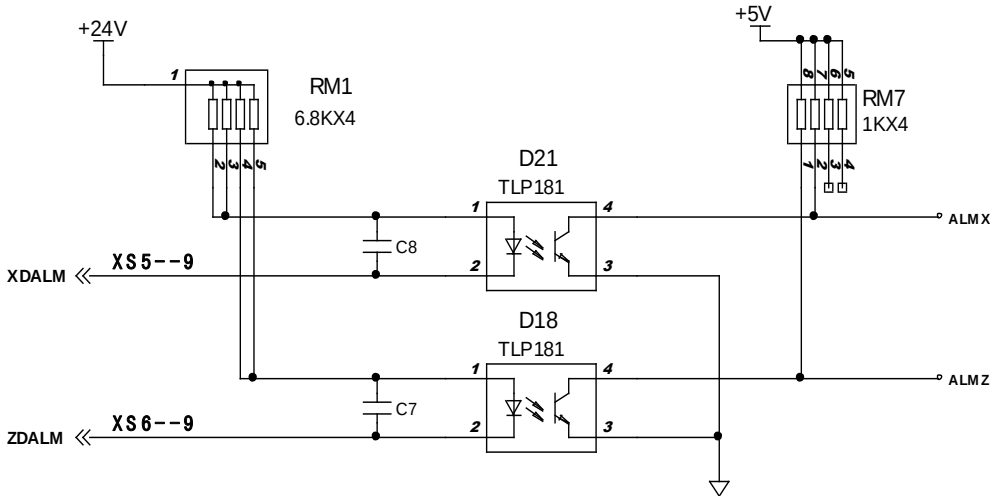
3.3.3.3 Alarm signal from driver (nDALM)

The effective level of the signal input into the system can be set as low level or high level effective through the parameter DALX/DALZ. This type of inputting circuit requires the driver side to provide signals in the following way:



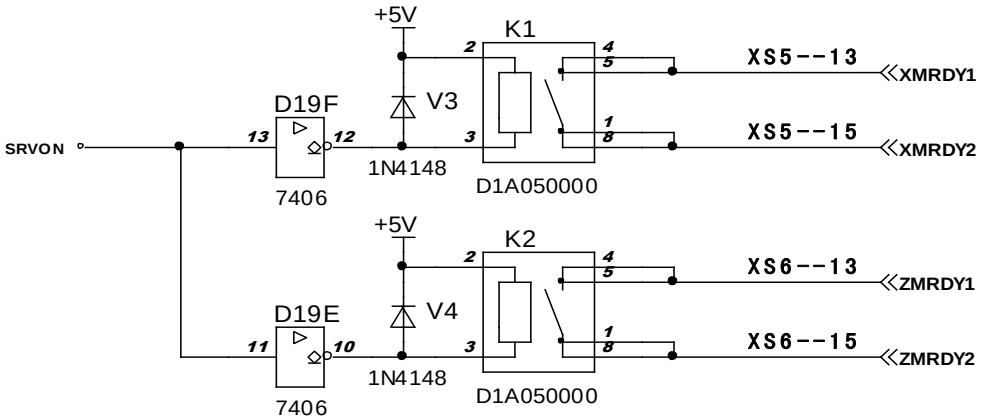
Way 1 (Level signal) Way 2 (Contact signal)

The receiving circuit at the system side for the signal is as follows:



3.3.3.4 CNC ready signals (nMRDY1/nMRDY2)

Diagram for the interface of the relay contact outputting circuit:



After the CNC system is initialized, the contact closes. If during the running, the CNC detects any driver alarm or emergency stop, the contact opens.

VP is a voltage (+5V or +24 V) supplied by the system to the driver, which is selected by SA6 setting switch. When 1-2 of SA3 are short circuited, VP=+24V, when 2-3 of SA3 are short circuited, VP=+5V.

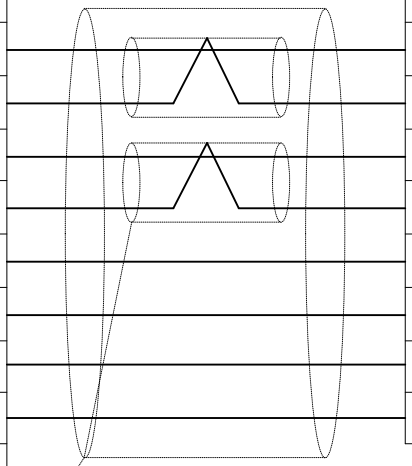
3.3.4 Definition of cables

Socket in the CNC side is DB15M.

3.3.4.1 Cable for BD3H/BD3L/BD3D/BD3S

XS51，XS52	
Signal	Pin
nCP	1
nCP-	9
nDIR+	2
nDIR-	10
nDALM	5
0V	14
+5V	13
nMRDY1	7
nMRDY2	8
0V	15

SIG X/Z	
Pin	Signal
7	CP+
8	CP-
11	DIR+
12	DIR-
19	RDY1
20	RDY2
32	EN+
31	EN-



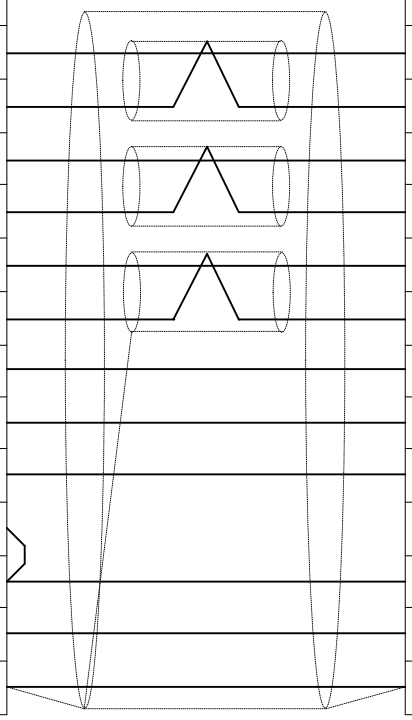
Welded on the metal body of socket

Line: RVVP 8 x 0.2 mm^2 (Pair-twisted shield cable)

3.3.4.2 Cable for Yaskawa AC servo drivers

XS51，XS52	
Signal	Pin
nCP	1
nCP-	9
nDIR+	2
nDIR-	10
nPC+	3
nPC-	11
0V	14
nALM	5
nMRDY1	7
nMRDY2	8
0V	15
VP = +24V	12
FG	壳

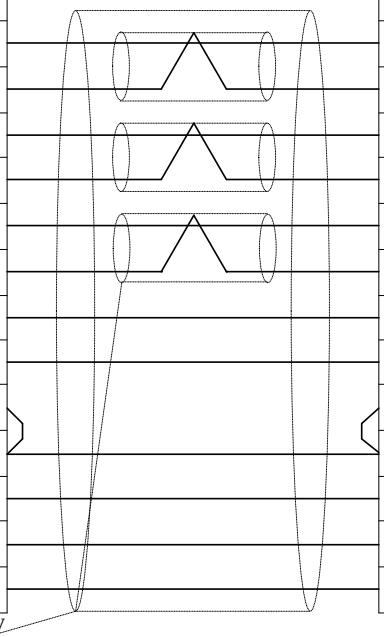
SIG X/Z	
Pin	Signal
7	PULSE
8	*PULSE
11	SIGN
12	*SIGN
19	PCO
20	*PCO
32	ALM-
31	ALM+
40	SV-ON
2	SG
47	+24V
50	FG



Line: RVVP 11 x 0.2 mm^2 (Pair-twisted shield cable)

3.3.4.3 Cable for Panasonic MINAS-A/A4 servo drivers

X51, X52	
Signal	Pin
nCP+	1
nCP-	9
nDIR+	2
nDIR-	10
nPC+	3
nPC-	11
nDALM	5
nMRDY1	7
nMRDY2	8
0V	14
0V	15
0V	15
VP = +24V	12



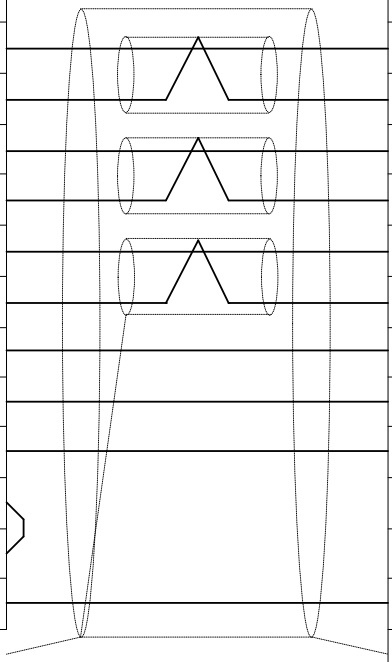
CN1 X/Z	
Pin	Signal
4	PULS2
3	PULS1
6	SIGN2
5	SIGN1
23	OZ+
24	OZ-
37	ALM+
29	SON
36	ALM-
13	GND
33	INH
41	COM-
7	COM+

Welded on the metal body of socket

Line: RVVP 12 x 0.2 mm^2 (Pair-twisted shield cable)

3.3.4.4 Cable for KND-SD100 servo driver

XS51,XS52	
Signal	Pin
XCP+	1
XCP-	9
XDIR+	2
XDIR-	10
XPC+	3
XPC-	11
0V	14
ALM	5
nMRDY1	7
nMRDY2	8
0V	15
VP = +24V	13



SIG X/Z：S360	
Pin	Signal
32	PULS+
33	PULS-
34	SIGN+
35	SIGN-
5	OZ+
6	OZ-
27	ALM-
26	ALM+
10	SON
18	COM+
36	FG

Welded on the metal body of socket

Line: RVVP 10 x 0.2 mm^2 (Pair-twisted shield cable)

3.3.4.5 Cable for KND SD200 servo driver

XS51,XS52			CN1 X/Z：DB44-3	
Signal	Pin		Pin	Signal
XCP+	1		14	PULS+
XCP-	9		13	PULS-
XDIR+	2		44	SIGN+
XDIR-	10		43	SIGN-
XPC+	3		5	OZ+
XPC-	11		21	OZ-
0V	14		8	ALM-
ALM	5		7	ALM+
nMRDY1	7		41	SON
nMRDY2	8			
0V	15			
VP = +24V	13		29	COM+
FG	Shell		15	FG

Line: RVVP 10 x 0.2 mm^2 (Pair-twisted shield cable)

3.4 Encoder Interface (XS53)

3.4.1 Signal list

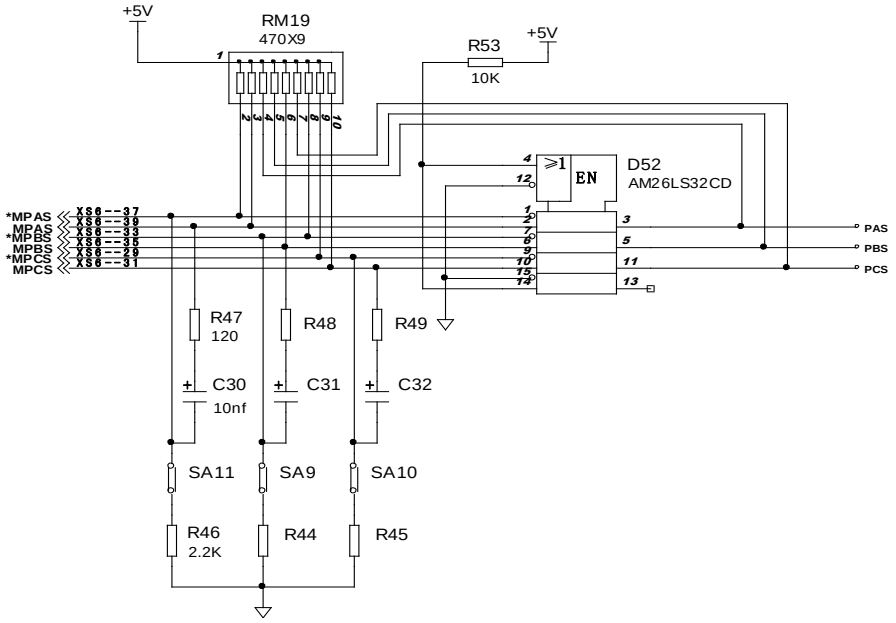
Socket model at the system side is: DB15F (DB type 15-pin female)

Plug model at the system side with welded cable is: DB15M (DB type 15-pin male)

XS53:DB15F（Spindle position encoder）

1	Keep unconnected	9	Keep unconnected
2	Keep unconnected	10	Keep unconnected
3	*MPCS	11	Keep unconnected
4	MPCS	12	+5V
5	*MPBS	13	+5V
6	MPBS	14	0V
7	*MPAS	15	0V
8	MPAS		

3.4.2 Circuit diagram



The receiving circuit is for standard connection. Unless otherwise specified, this circuit will be employed as default for the CNC system. Model of the spindle position encoder with KND is LF-102.4BM-C05D, pulse per revolution is 1024, work voltage is +5V (from Changchun First Optic Instrument Factory).

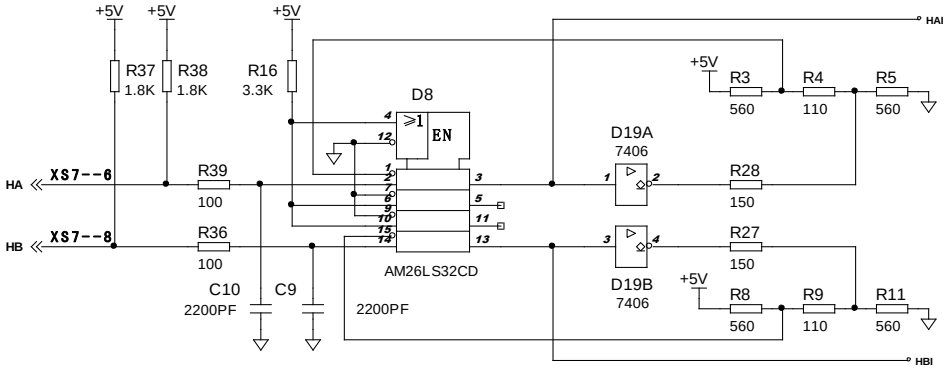
3.4.3 Cable definition

XS53: DB15 M		Encoder	
Signal	Pin	Pin	Signal
*MPCS	3		*PCS
MPCS	4		PCS
*MPBS	5		*PBS
MPBS	6		PBS
*MPAS	7		*PAS
MPAS	8		PAS
5V	12		5V
0V	14		0V
Welded on the metal body of socket			FG

Line: RVVP10 x 0.2 mm^2 (Pair-twisted shield cable)

3.5 Interface of Handwheel (XS55)

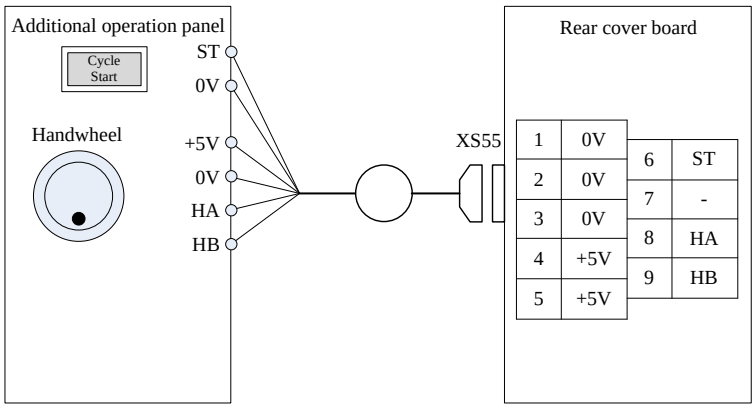
3.5.1 Circuit diagram



3.5.2 Connection diagram

Socket model at the system side is: DB9F (DB type 9-pin female)

Plug model at the system side with welded cable is: DB9M (DB type 9-pin male)



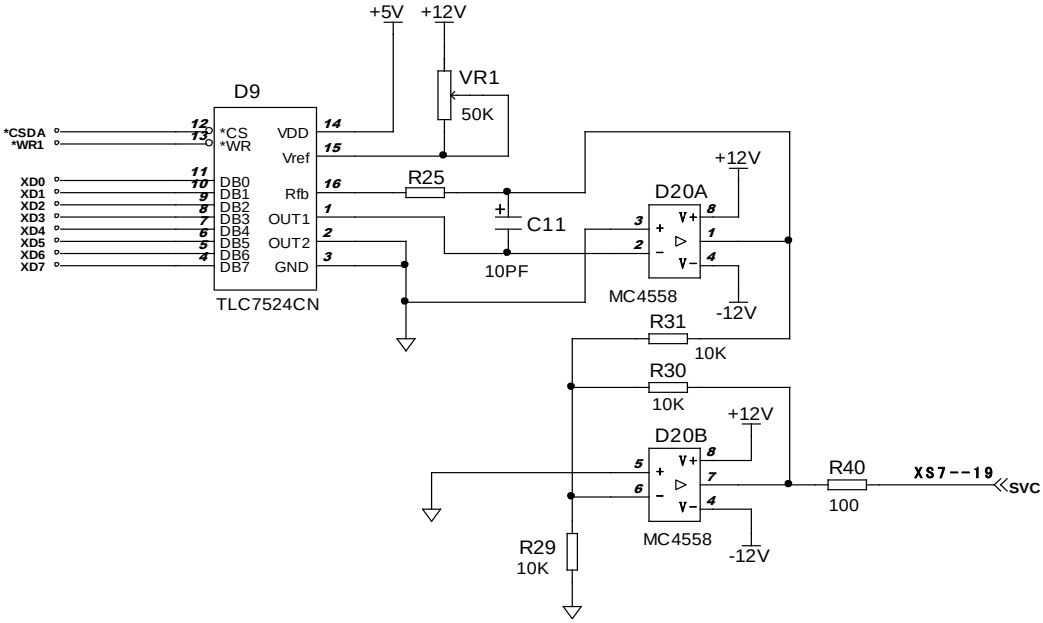
The model of the manual pulse generator with KND is LGF-001-100, the pulse per revolution is 100, and the operation voltage is +5V.

NOTE

The signal shall be transmitted as far as possible by a twisted shielded cable.

3.6 Analog Spindle Interface (XS56)

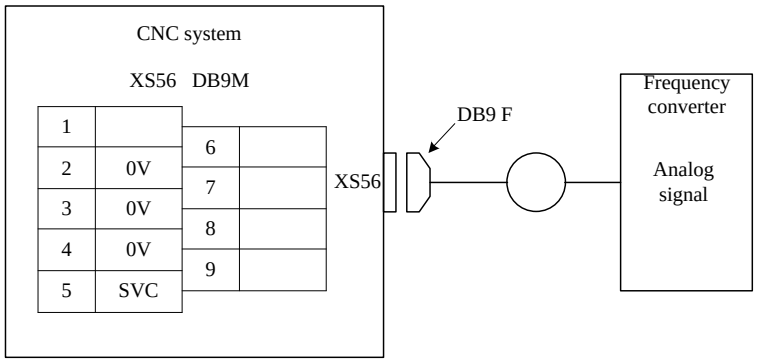
3.6.1 Circuit diagram



3.6.2 Connection diagram

Socket model at the system side is: DB9M (DB type 9-pin male)

Plug model at the system side with welded cable is: DB9F (DB type 9-pin female)

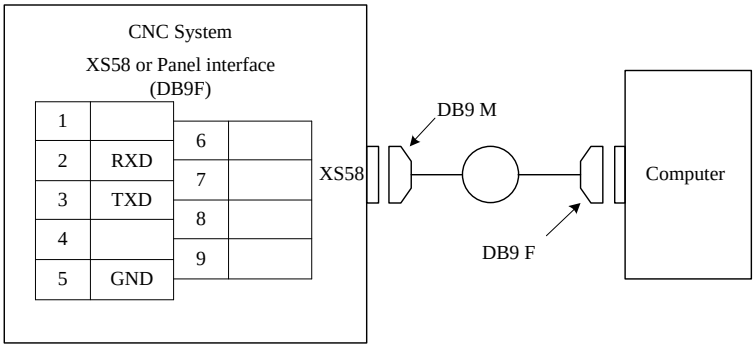


NOTE

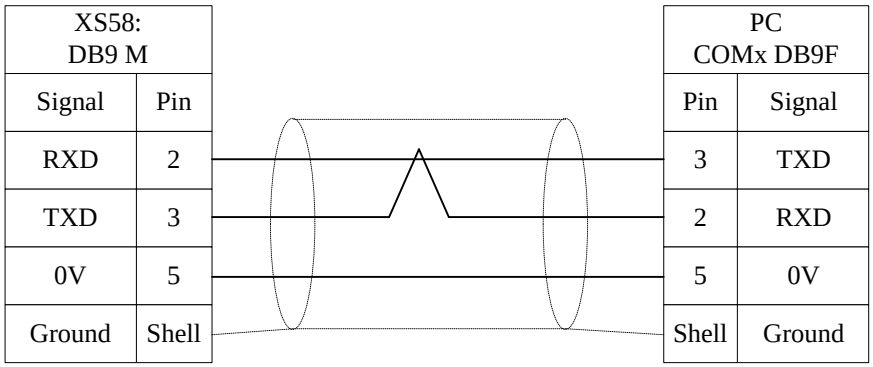
The signal shall be transmitted as far as possible by a twisted shielded cable of RVVP2X0.5mm^2.

3.7 RS232C Interface

3.7.1 Connection diagram



3.7.2 Cable definition

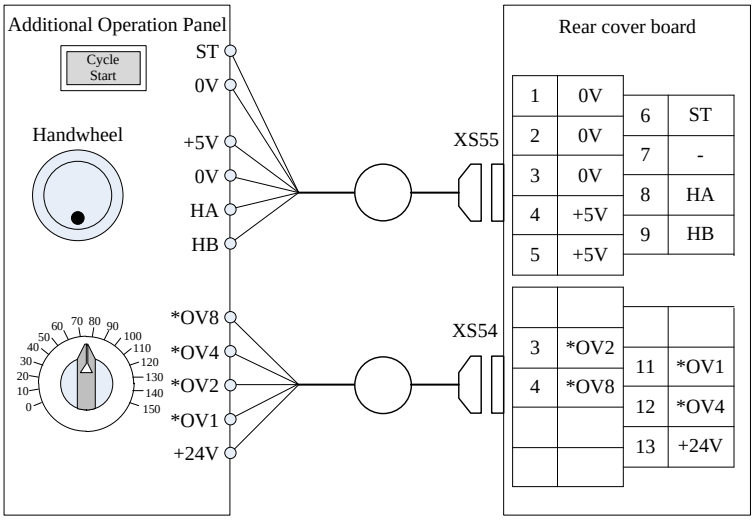


XS58: DB9 M		PC COMx DB9F	
Signal	Pin	Pin	Signal
RXD	2	3	TXD
TXD	3	2	RXD
0V	5	5	0V
Ground	Shell	Shell	Ground

Line: RVVP 3 x 0.3 mm^2 (Pair-twisted shield cable)

3.8 Connection of Additional Operator's Panel

3.8.1 Connection diagram

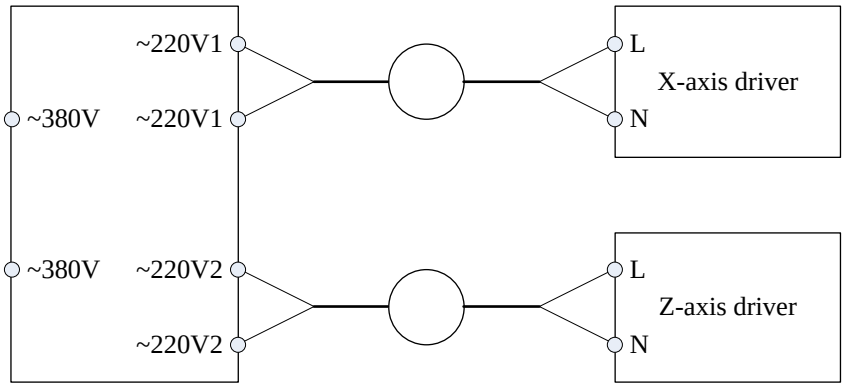


As for the circuit of the handwheel interface, please refer to section V3.5.

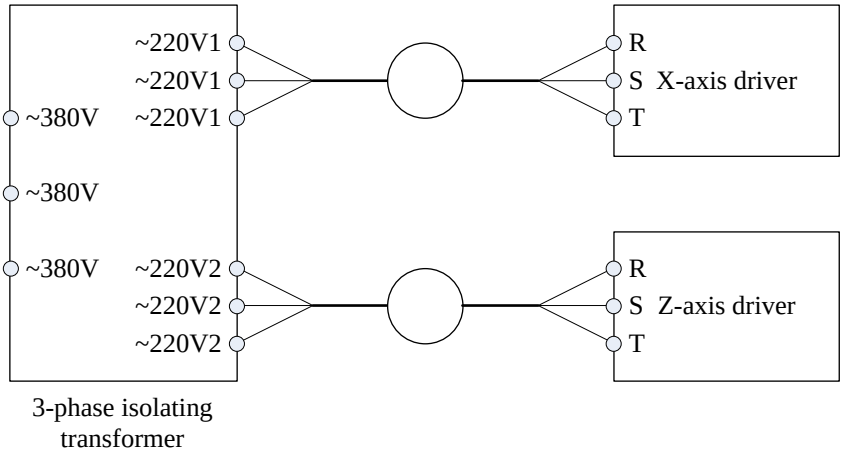
As for the circuit of the override switch interface, please refer to section 5.4.5.

3.9 Connection of Isolating Transformers

3.9.1 Connection diagram for Step motor drivers (BK-1.3)



3.9.2 Connection for AC servo drivers (SSG-3/0.5)



4 MACHINE INTERFACE

4.1 Input Signals

4.1.1 DC input signal A

The DC input signals A are signals from the machine to the CNC system, which are from the keys, limit switches, contacts of the relays and the proximity switches at the machine side.

The contact at the machine side shall meet the follows:

- Contact capacity: DC30V and 16mA above.
- The leakage current between contacts at an open circuit shall be less than 1mA (Vmax＝26V).
- The voltage drop between contacts at closed circuit shall be less than 2 V (current 8.5 mA, incl. the voltage drop from the cable).

As for the circuit of the signals, refer to Figure V-7

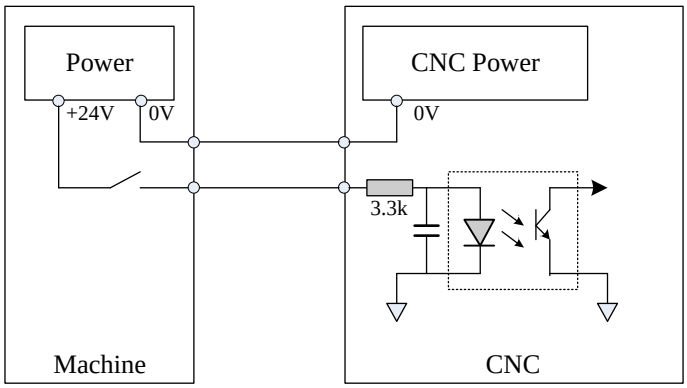


Figure V-7

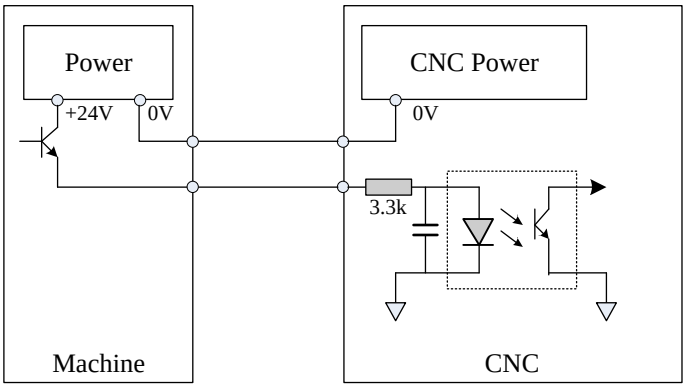
4.1.2 DC input signal B

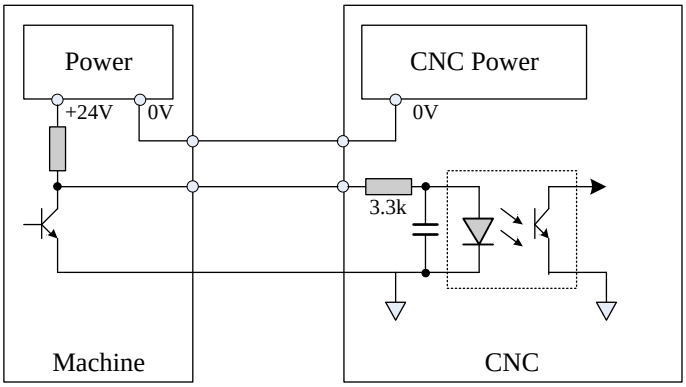
The DC input signals B are signals from the machine to the CNC system, and are signals used at high speed.

The contact at the machine side shall meet the follows:

- Contact capacity: DC30V and 16mA above.
- The leakage current between contacts at open circuit shall be less than 1mA (Vmax＝26V).
- The voltage drop between contacts at a close circuit shall be less than 2 V (current 8.5 mA, incl. the voltage drop from the cable).

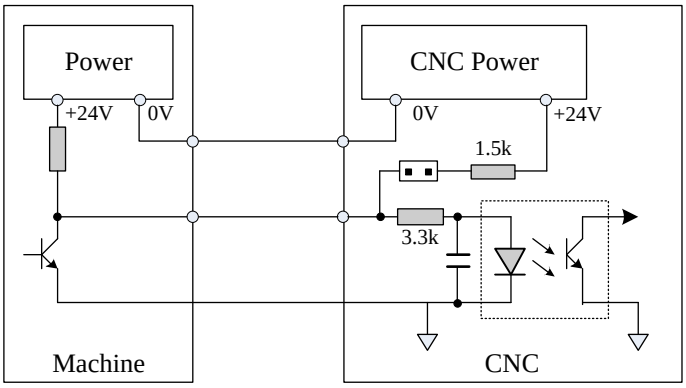
As for the circuit of the signals, refer to figures below.





4.1.3 Input signal with interior pull-up resistors

The tool rest signal T01~T08 and home deceleration signal *DECX and *DECZ are furnished with interior pull-up resistance, and are easy to connect with NPN or PNP Hall switch. The circuit of the main board is as follows:



When the socket for the circuit of the corresponding input end on the main board is short circuited, it has a pull-up resistance of 1.5k in the input interface, it is possible to directly connect the NPN Hall switch or input a low effective level.

When the socket for the circuit of the corresponding input end on the main board is not short circuited, the input interface can be directly connected with the PNP Hall switch or can input a high effective level.

In the default settings for the system, the tool rest signals (T01~T04) are so set that they have a pull-up resistance (the corresponding socket on the main board is short circuited), in this way, the system can be directly connected with the tool rest that employs a NPN Hall switch (e.g. Changzhou Hongda Tool rest).

The rest input signals (T05~T08, *DECX and *DECZ) are so set that they have no pull-up resistance (the corresponding socket on the main board is not short circuited).
Refer to the diagrams above to adjust machines according to your actual applications.

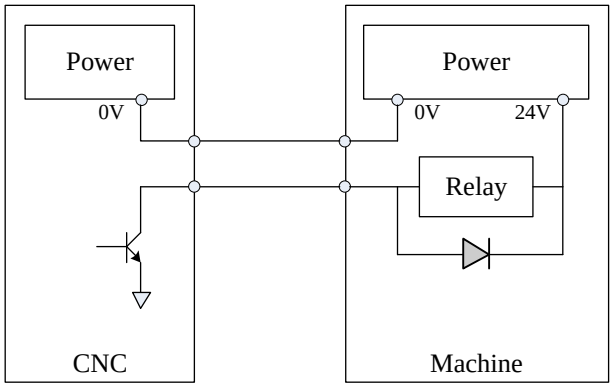
4.2 Output Signals

DC output signals are for driving the relays and indicator lamps at the machine side. The output circuit of the system is of Darlington tube.

4.2.1 Attributes about Darlington tube output signals

1 The max. load current when the output is ON is 200 mA below (incl. instantaneous current).
2 The saturation voltage is corresponding maximal 1.6V at 200 mA when the output is ON, and the typical value is 1 V.
3 The withstanding voltage when the output is OFF is 24V＋20％ below (incl. instantaneous voltage).
4 The current leakage when the output is OFF is 100μA below.

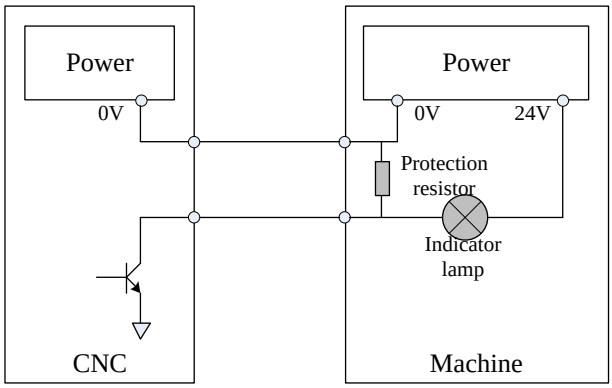
4.2.2 Output signal to drive relay circuit



NOTE

When an inductive load(relay etc.)is connected at the machine side, a spark suppressor must be used. And the spark suppressor shall be located as near as possible to the load (within 20 cm). When a capacitive load is connected at the machine side, a current-limiting resistance must be connected in series, the voltage and current of the load incl. the instantaneous values must be within the ratings.

4.2.3 Output signal to drive indicator lamp



NOTE

When lighting the indicator lamp directly using the transistor output, shock current will generate, which is highly liable to damage the transistor. Therefore, you have to design a protective resistance as shown in the above figure. Its voltage and current incl. the instantaneous values must be within the ratings.

4.3 I/O Signal List

4.3.1 Input signals' list

DGN.0	7	6	5	4	3	2	1	0
Pin			XS50:21	XS50:8				
Signal			*DECX	QPI				

DGN.1	7	6	5	4	3	2	1	0
Pin			XS50:20	XS50:7	XS50:19	XS50:6	XS50:18	XS50:5
Signal			*DECZ	*ESP1	T04	T03	T02	T01

DGN.2	7	6	5	4	3	2	1	0
Pin					XS2:5	XS2:4	XS2:3	XS55:6 XS2:2
Signal					*ESP2	*SPL	*SPK	ST

DGN.3	7	6	5	4	3	2	1	0
Signal	X37	X36	X35	X34	X33	X32	X31	X30
Pin	XS54:4	XS54:12	XS54:3	XS54:11	XS54:2	XS54:10	XS54:1	XS54:9
Signal	*OV8	*OV4	*OV2	*OV1	T08	T07	T06	T05
			M42I	M41I	M93I	M91I	M23I	M21I
	/*LMZ	/*LPZ	/*LMX	/*LPX	*LMZ	*LPZ	*LMX	*LPX

	QPJI	QPSI	RHI		BDT	SAGT *SPK2	TWI	

1 *OV8～*OV1: External override signals. It is available when the total tool number is less than 8 and SOVI is set to 1.

2 M41I/M42I: Spindle gear feedback signals.

3 M93I/M91I/M23I/M21I: Input signals used by special M codes described in section II5.11 and II5.12

4 /*LMZ~/*LPX: Input signals used by hardware stroke limitation function. They are available when MOT is set to 0 and LPMH is set to 1.

5 *LMZ~*LPX: Input signals used by hardware stroke limitation function. They are available when both MOT and LPMH are set to 0.

6 QPJI/QPSI: Chuck Clamped In Position signal, which is available when QPIN set to 1.

7 TWI: Tailstock control signal, which is available when TWSL is set to 1.

8 BDT: Input signal for controlling ON/OFF of the optional block skip function, which is available when SBDT is set 1.

9 *SPK2: External “Feed hold” signal.

10 RHI: Alarm signal indicating that oil level is too low used by lubrication.

4.3.2 Output signals’ list

DGN.4	7	6	5	4	3	2	1	0
Pin	XS50:1	XS50:14	XS50:2	XS50:15	XS50:3	XS50:16	XS50:4	XS50:17
Signal	SPZD	TL-	TL+	M32	M08	STL	M04	M03
						SAGTO		

SPZD : Spindle brake signal

TL+/TL- : CCW & CW signal for tool rest

M32 : Lubrication signal

M08 : Coolant signal

M03 : CW rotation signal of spindle

M04 : CCW rotation signal of spindle

STL : Automatic running indicator signal

SAGTO : Protective gate signal

DGN.5	7	6	5	4	3	2	1	0
Pin	XS57:3	XS57:5	XS57:6	XS57:8	XS57:7	XS57:4	XS57:2	XS57:1
Signal	QPJ	QPS	Y25	ESP0	S04	S03	S02	S01
			M23O	M21O			M42O	M41O
			TWJ	TWT				

QPJ/QPS : Chuck clamping/loose signal

ESP0 : Emergency stop signal

S01~S04 : Spindle gears’ signal

M23O/M21O : Output signals for special M codes

M41O/M42O : Output signals of high and low gear used by analog spindle

TWJ/TWT : Output signals controlling tailstock to move forward and backward. It is available when TWSL is set to 1

NOTE

TWJ/TWT and M23O/M21O use the same output terminals. Bit parameters TWSL and M21O/M23O should not be set to 1 at the same time, or system will clear them to 0 automatically.

4.3.3 I/O signals' arrangement (XS50 & XS54)

I/O signals' arrangement in XS50 (DB25F)

1	SPZD	14	TL-
2	TL+	15	M32
3	M08	16	STL
4	M04	17	M03
5	T01	18	T02
6	T03	19	T04
7	*ESP1	20	*DECZ
8	QPI	21	*DECX
9	0V	22	+24V
10	0V	23	+24V
11	0V	24	+24V
12	0V	25	+24V
13			

Additional input signals' arrangement in XS54 (DB15M)

1	T06(X31)	9	T05(X30)
2	T08(X33)	10	T07(X32)
3	*OV2(X35)	11	*OV1(X34)
4	*OV8(X37)	12	*OV4(X36)
5	0V	13	+24V
6	0V	14	+24V
7	0V	15	+24V
8	0V		

Additional output signals' arrangement in XS57 (DB15M)

1	S01	9	0V
2	S02	10	0V
3	QPJ	11	0V
4	S03	12	0V
5	QPS	13	+24V
6	Y25	14	+24V
7	S04	15	+24V
8	ESPO		

1 When SOVI is set to 1, #3, #4, #11 and #12 terminals in XS54 are used as external override signals (*OV8~*OV1).

2 When MOT is cleared to 0 and LPMH is set to 1, X37~X34 are used as hardware stroke limitation signals.

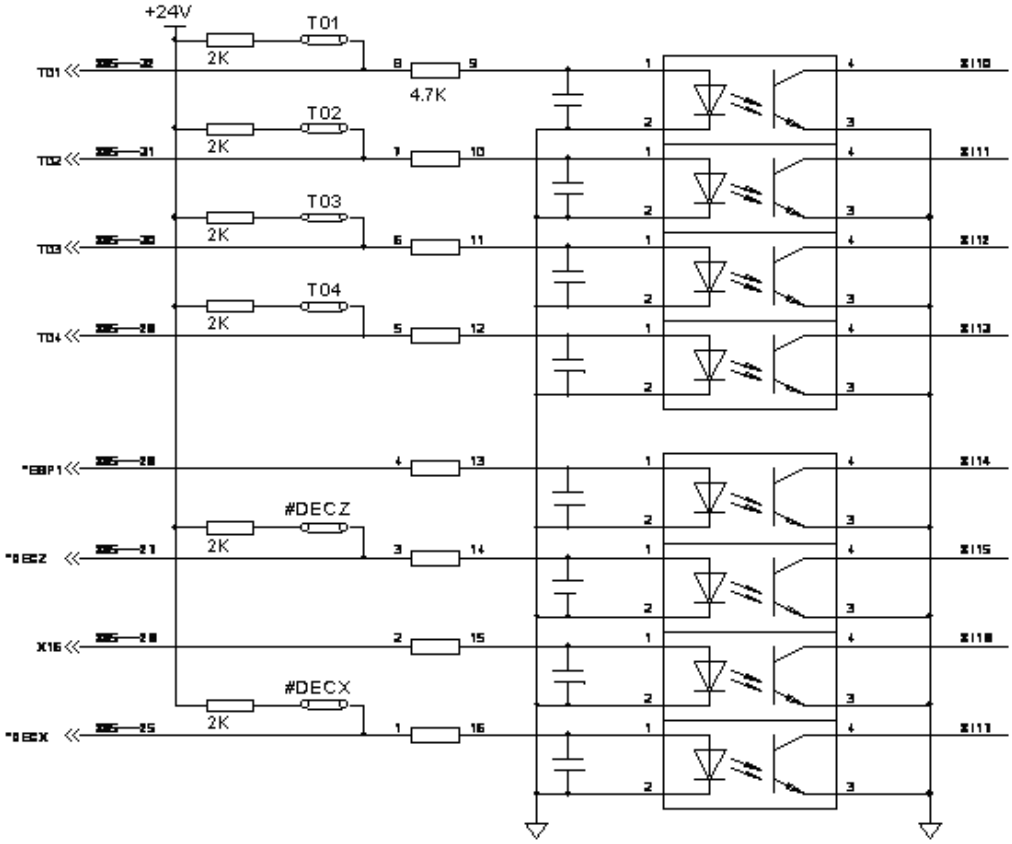
3 When both MOT and LPMH are cleared to 0, X33~X30 are used as hardware stroke limitation signals.

4 When MOT is set to 1, hardware stroke limitation function is invalid.

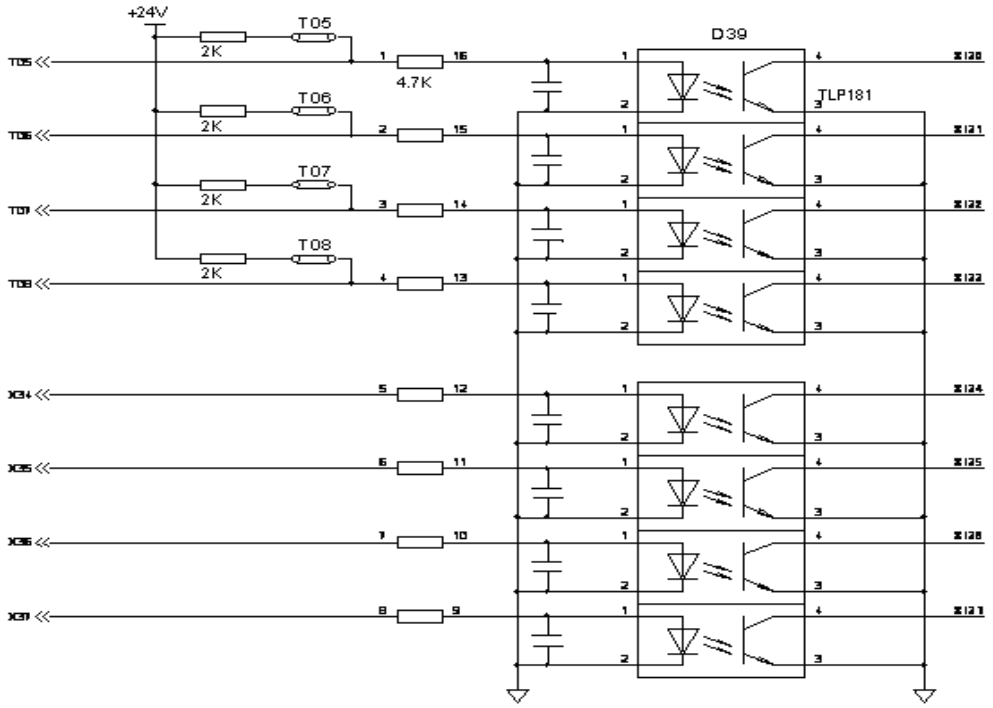
4.3.4 I/O signals' circuits

4.3.4.1 Input signals' circuits

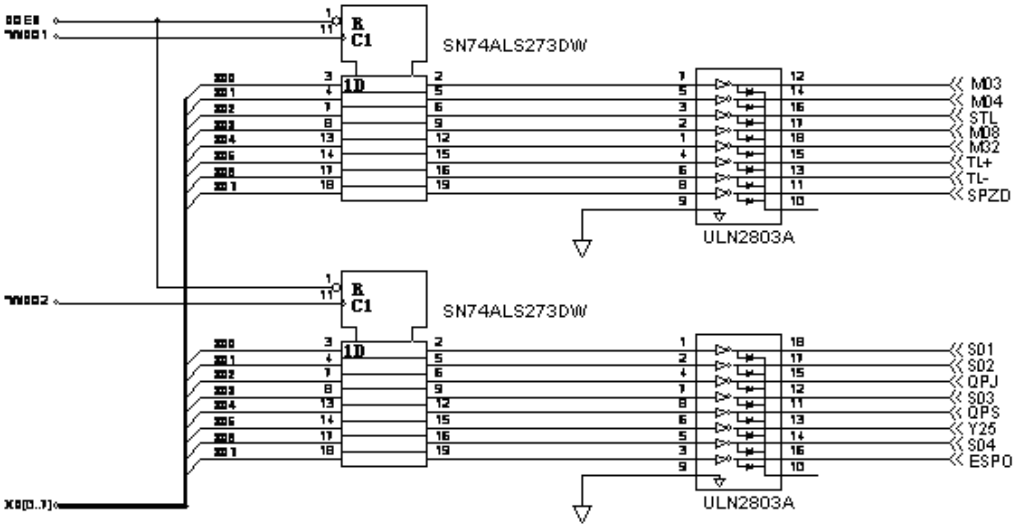
Input signals' circuits (XS50)



Additional input signals' circuits (XS54)



Output signals' circuits



4.4 Description of I/O Signals

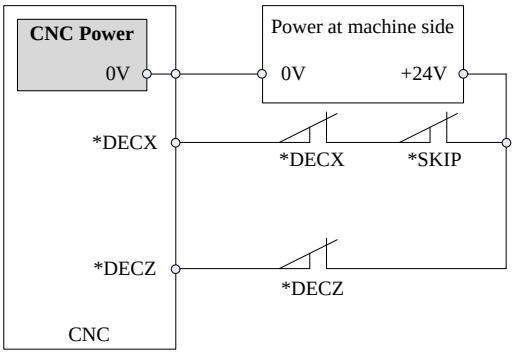
4.4.1 Input signals

*DECX & *DECZ

The signal is used when returning to the mechanical reference point, and is of normal close contact. The process for returning to the reference point is:

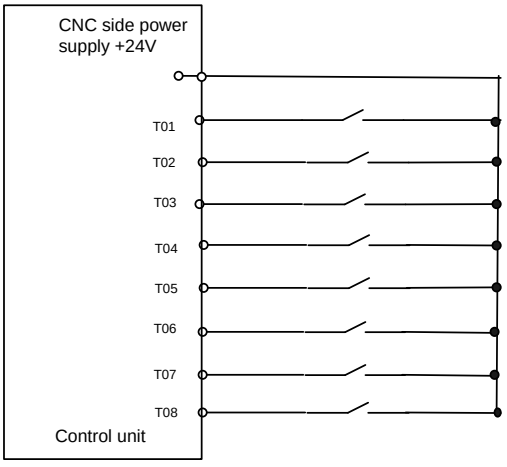
Select the mechanical homing way, then press the manual feed key of the corresponding axis, now the machine will move with the rapid rate to the reference point. When the deceleration signal for returning to the reference point (*DECX and *DECZ) contact is open (the deceleration switch is pressed), the feed rate will decrease immediately, then the machine will continue to move at constant low speed. When the deceleration switch is released, the deceleration signal contact re-closes, then the system detects the one-rotation signal of the encoder or the magnetic switch signal (PC signal). If the signal changes from high level to low level (the drop edge of the PC signal is detected), the movement stops, meanwhile, the machine coordinates are zeroed, and the returning to the reference point completes. The manual feed hold is invalid until the Homing mode is cancelled.

The connection diagram is as follows:

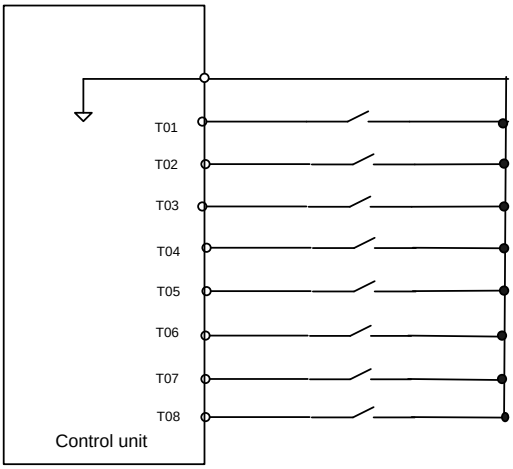


T01～T08

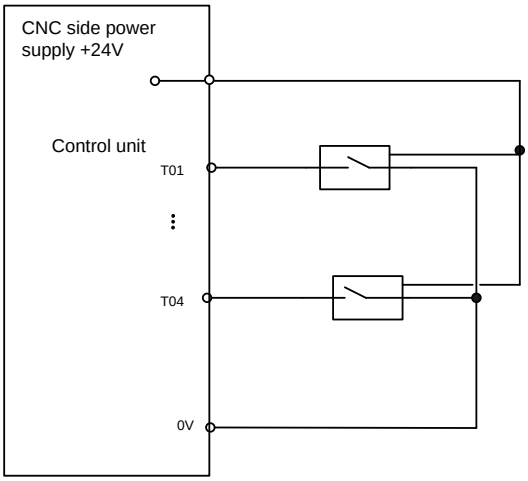
The effective level is High. When one signal among them is of high level, it means the tool rest is located at the position of the tool no. The connection when no interior pull-up resistance is used is as follows (the corresponding setting switch is set as not short circuited):



The connection when an interior pull-up resistance is used is as follows (the corresponding setting switch is set as short circuited):



When the system is used with a Changzhou tool rest, wiring can be done as follows (the interior pull-up resistance of the corresponding signal is set as effective):



During the tool changing, after the tool rest is in position, the tool rest CW output signal (TL+) becomes off, and the system outputs the tool rest CCW output signal (TL-) after delays a time set by parameter 037, the width for the signal is the time set by parameter 038; then the system turns off the tool rest CCW output signal (TL-), and the T code command ends. The program proceeds the next block.

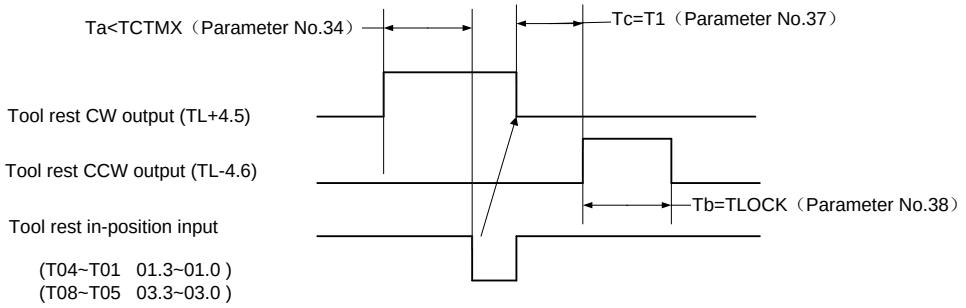
At the initialization when power ON, the initial values of 034, 037, 038 and 039 are as follows:

Parameter No.	Meaning	time	Def. value
034	Max. time for tool rest CW TCTMX (limit time for tool changing)	80 sec	80000 (ms)
037	Delay time from the end of tool rest rotation CW to the start of rotation CCW T1	0.5 sec	496 (ms)
038	Tool rest rotation CCW locking time TLOCK	0.5 sec	496 (ms)
039	Selection of total tool positions	4 (unit: no. of tool pockets)	

At left lower corner of the screen, T indicates the T code and tool offset no. of current command. At power on, T code means the last tool no. When a tool change ends normally, the system automatically modifies the value. After commanding a T code, if the tool rest is not in place due to some reason, T shows the tool no. that is before the tool change. The system will not change a tool when the commanded tool no. is same as the displayed tool no.

For a manual tool change, the T code will change to a new value only after the tool change ends.

The time sequence for tool changing is show below:



TCTMX represents the longest time tool changing is performed successfully.

When Ta is longer than TCTMX, ALARM (№05) occurs

Active level of tool rest in-position signals (T08～T01) is set by TSGN:

TSGN=0: Logic high is active. (normal open)

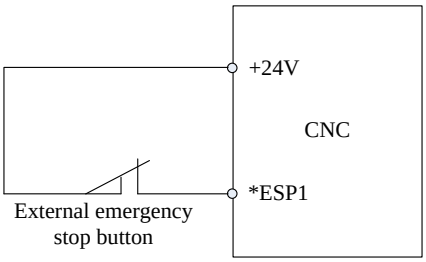
TSGN=1: Logic low is active. (normal close)

*ESP1: Emergency stop signal

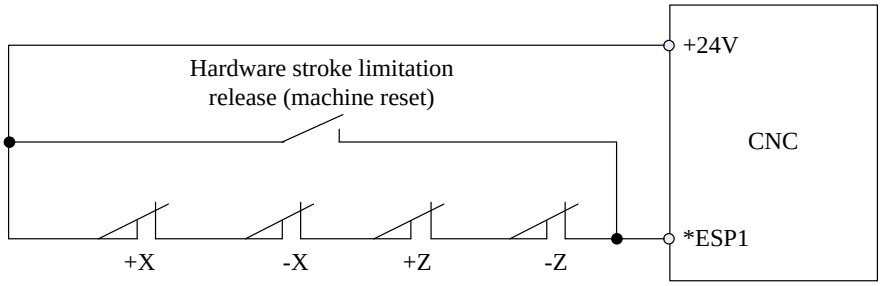
When bit MOT of parameter P001 is set as 1, the input signal *ESP1 is used as the emergency stop signal for the external input (function same as emergency stop switch signal of the system's panel *ESP2).

The signal is a normal-close contact signal. When the contact is open, the control system resets, and the machine emergency stops. After emergency stop, the system ready signal MRDY will be off. Meanwhile, the outputting of the movement command will be blocked.

When this function is not required, you can disable it by setting the bit MESP of parameter P001 as 1.



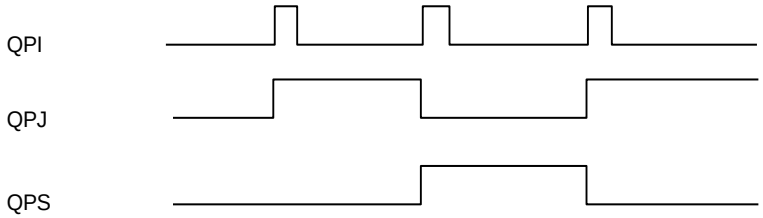
If the input signal of the system X37-X30 are defined as other functions and it is impossible to provide hard limit input, you can use *ESP1 as hard limit input signal. The connecting diagram is as follows.



QPI: Chuck (foot switch) signal

The time sequence for the chuck movement is as follows (taking as an example the output of level

signal):



When power ON, both the output signals Chuck Clamped QPJ and Chuck Unclamped QPS are zero.

When the spindle starts CW/CCW, the chuck must be clamped. If setting QPIN=1 for system parameter 043, you shall also check the Chuck Clamped In Position signal. Otherwise, the system will generate ALARM (№15): "Spindle is started with unclamped chuck".

When the spindle is rotating or braking from rotation to stop, the foot chuck switch is invalid.

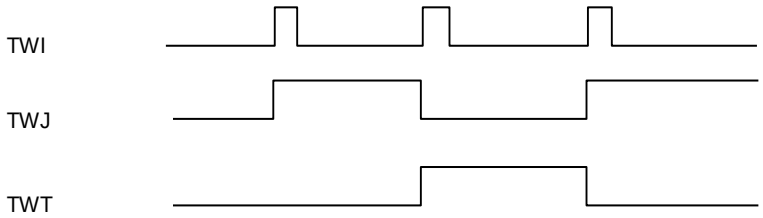
When parameter QPLS=1, the chuck output is of pulse output, and the pulse width is set by parameter P51 in ms.

In "Test" screen, you can select Inner/Outer Chuck.

You can control the tightness of the chuck by program code M10/M11. For details, refer to 2.5.11.

TWI: Tailstock (foot switch) signal

The time sequence diagram for the tailstock action is as below:



*ESP2

This is the emergency stop (ES) signal from the panel switch, which is the 2nd ES signal input point of the CNC system. You can disable it by setting MSPL of the parameter 001 as 1.

*SPK: Feed hold signal

This is the feed hold signal from the 3-position switch on the panel when it is placed at middle position. You can disable it by setting MSPL of the parameter 001 as 1.

*SPK2: Feed hold signal 2

This is the feed hold signal when the user configures the machine panel by himself (button switch). You can disable it by setting SPK2 of the parameter 042 as 1. When SPK2=1, the external pause switch control is effective (note: now the pause switch on the panel is still active). In this case, if the 3-position switch on the panel is placed at left place, pressing the external Pause button, the system will pause; and now if you press Cycle Start button, the system will continue to run. If the 3-position switch on the panel is placed at middle or right side, the system will be paused; when SPK2=0, the external Pause button is invalid, only the Pause switch on the panel is active.

ST: Cycle Start signal

It is the Cycle Start signal from the button switch of the panel.

*SPL: Spindle Pause signal

This is the Spindle Pause signal from the 3-position switch on the panel when it is placed at 3# position. You can disable it by setting MSPL of the parameter 001 as 1.

*OV8～*OV1

Override switch signal from the additional panel.

M93I, M91I

User interface GO TO function input signal (refer to section 2.5.16).

M23I, M21I

Special M code input signal (refer to section 2.5.15).

/*LMZ, /*LPZ, /*LMX, /*LPX, *LMZ, *LPZ, *LMX, *LPX

Input signal for hard limit. In addition to soft limit by parameters, it is also possible to limit the +, - direction of the axes by inputting signals. When an axis is under + direction limit, the axis can be moved manually only in reverse direction.

- **Parameter**

001		MOT						

6 MOT Mask the hardware travel limitation switches

0: Not mask

1: Mask

MOT doesn't affect the stored stroke limitation function. The stored stroke limitation function is disabled by setting parameters P15~P18 to zero.

041								LPMH

0 LPMH Hardware stroke limitation switches' source selection

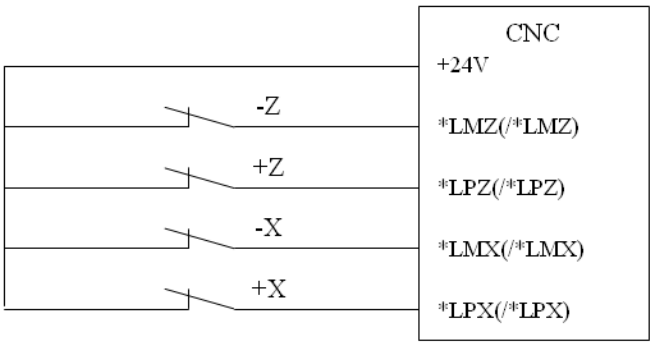
0: Lower 4-bit in DGN.003 are used as hardware stroke limitation switches

1: Higher 4-bit in DGN.003 are used as hardware stroke limitation switches

- **Input signals**

DGN.3	/*LMZ	/*LPZ	/*LMX	/*LPX	*LMZ	*LPZ	*LMX	*LPX
Interface	XS54:4	XS54:12	XS54:3	XS54:11	XS54:2	XS54:10	XS54:1	XS54:9

- **Wiring diagram**



BDT

Control the optional block skip function effective or not.

SAGT: Input signal for protective gate function

- **Function of protective gate:**

  In AUTO mode, if you start a program or a program is running when the protective gate is open, the alarm "PROTECT GATE IS OPEN" will occur.

- **The controlling functions for the protective gate are as follows:**

  1 In any mode, when the protective gate is open:
  - The output signal for M03/M04 will be off and cleared
  - The output signal for S01~S04 will be off and cleared; the voltage for the analog spindle will be set to zero.
  - The axis movement will stop.
  - It will output "PROTECT GATE IS OPEN"

  2 When the protective gate is open under AUTO mode:
  - If attempting to start a program, the operation will be invalid, at the same time, the alarm will occur besides the processing in the first item above. Pressing "RESET" can release the alarm.
  - If no attempt to start program, only processing as shown in the first item above will occur.

  3 In MDI mode, when the protective gate is open, the system program can be executed normally.

  4 When the protective gate is open:
  - The code S01~S04 can't be executed, S simulation commands can't be executed.
  - The code M03/M04/M05 can be executed.

- **Parameter**

036							SSGT	SAGT

1 SSGT Select the GATE PROTECTION function

0: Invalid

1: Valid

0 SAGT Select the working style of GATE PROTECTION function

0: The logic high state of the input signal indicates the gate is closed

1: The logic low state of the input signal indicates the gate is open

- **I/O signals**

DGN.3						SAGT		

SAGT: external input signal for protective gate

Some input signals support multiple functions. Set corresponding parameters for using these signals according to your actual applications.

DGN.4						SAGTO		

SAGTO: the output signal for protective gate

RHI

Input signal indicating that the lubrication oil level is low.

4.4.2 Output signals

All output signals of the system are supplied by Darlington tubes. When the output is active, the corresponding Darlington tube is on, which is equivalent to that the signal end of the external load is connected with 0 V. The common end of all external loads is +24V supplied by the user.

TL＋, TL－

Tool rest control signals. TL＋ is for tool rest CW rotation, TL- is for tool rest CCW rotation.

M03, M04, M05

Spindle control signals. M03 is for spindle CW rotation, M04 is for spindle CCW rotation, M05 is for spindle stop rotation.

M08

Coolant control signal. M08 turns on coolant and M09 turns off coolant.

M32

Control lubrication of the machine. M32 turns on lubrication and M33 turns off lubrication.

SPZD Spindle Brake signal

SPZD is used to brake spindle. The control logic is associated with the ZDT1S parameter.

044				ZDT1S				

4 ZDT1S Select the work style of the delay time 1 when brake spindle

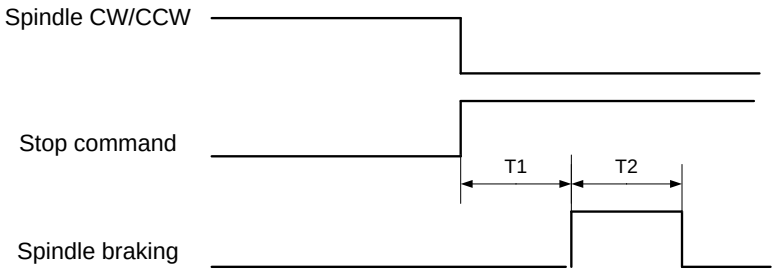
0: T1 is specified by parameter

1: T1 is fixed as 0 when analog spindle is enabled.

NOTE

When SANG2 is set to 1, the function is invalid. T1 is specified by parameter.

The control logic is as follows:



T1: When the spindle is running, after sending Spindle Stop (auto or manual) command, spindle CW/CCW rotation will be turned off immediately. The Spindle Brake signal is sent after delaying T1 time (0.5 sec). The time of T1 is fixed to be 0.5 sec by the system.

T2: time for spindle braking, which is set by no.040 parameter.

ESPO

ESPO indicates emergency stop or some alarm happens. It is available when M21O is cleared to 0.

Bit parameter ESPO in P42 controls the working style of ESPO output signal:

ESPO=0: System output ESPO signal only when emergency stop or some driver alarm happens.

ESPO=1: System output ESPO signal when any alarm happens.

S01~S04

Signals to adjust spindle speed..

STL

In auto running mode, system outputs STL signal.

VI APPENDIX

1 PARAMETERS

1.1 Description In Sequence

001	MSPL	MOT	MESP	SINC	CPF4	CPF3	CPF2	CPF1

[Def. value] 01001100

7 MSPL Mask the signals from panel including *ESP2, *SPL and *SPK

0: Not mask

1: Mask

6 MOT Mask the hardware travel limitation switches

0: Not mask

1: Mask

5 MESP Mask the signal *ESP1

0: Not mask

1: Mask

4 SINC Mask the incremental steps 0.1 and 1 used in HANDLE and SINGLE-STEP mode

0: Not mask

1: Mask

3 CPF4 Specify the pulse frequency for pith error and backlash compensation

2 CPF3 Specify the pulse frequency for pith error and backlash compensation

1 CPF3 Specify the pulse frequency for pith error and backlash compensation

0 CPF1 Specify the pulse frequency for pith error and backlash compensation

Available for all axes. Fre.= (Setting value + 1) kpps

002	ZRSZ	ZRSX	ZCZ	ZCX	DALZ	DALX	DIRZ	DIRX

[Def. value] 00000000

7 ZRSZ Setting the mechanical zero point of Z-axis

0: No mechanical zero point (Type A reference position retuning)

1: Having mechanical zero point (Type B reference position retuning)

6 ZRSX Setting the mechanical zero point of X-axis

0: No mechanical zero point (Type A reference position retuning)

1: Having mechanical zero point (Type B reference position retuning)

5 ZCZ Deceleration switch and zero-point signal are needed or not when returning the reference position (Z-axis)

0: Need

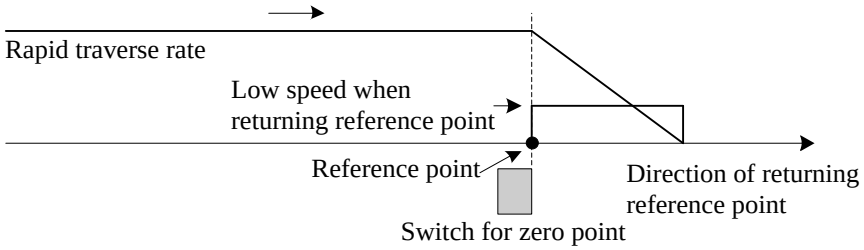
1: Type C reference position retuning

4 ZCX Deceleration switch and zero-point signal are needed or not when returning the reference position (X-axis)

0: Need the two signals.

1: Type C reference position retuning

NOTE



Using the type C to return the reference position, it is necessary to set ZMZ and ZMX in P004 to select the reverse direction of the returning movement.

3 DALZ Select the active level of alarm signal from driver of Z-axis

2 DALX Select the active level of alarm signal from driver of X-axis

1 DIRZ Select the rotation direction of the motor driving Z-axis

0 DIRX Select the rotation direction of the motor driving X-axis

003	BDEC	BD8	RVDL	SMZ	KSGN	ZNIK	TSGN	ABOT

[Def. value] 00001110

7 BDEC Select the style of backlash compensation

0: Type A, Output pulses at a fixed frequency (specified by CPF4, 3, 2, 1 and BD8)

1: Type B, Output pulses at an acceleration or deceleration speed

6 BD8 Select the pulses' frequency used by backlash compensation

0: At the frequency specified by CPF4, 3, 2, 1

1: At the 1/8 times of the frequency specified by CPF4, 3, 2, 1

NOTE

Rebooting is needed to activate the setting of BD8.

5 RVDL Select the working style of the SIGN and PULSE signals outputting when changing the movement direction

0: SING and PULSE signals output at the same time

1: Output the PULSE signal after a certain time of the SIGN signal outputs

4 SMZ Select the control style of cutting speed at the corner between two or more blocks

0: Apply a special control method at the corner

1: Axes stop before executing the next movement instruction

3 KSGN Weather keep the button state when moves in the negative direction

0: Don't keep

1: Keep

2 ZNIK Weather keep the state of axis control button or not when returning the reference position

0: Not keep

1: Keep. Axis stops when reaching the zero point. Press [RESET] if necessary to stop the movement

1 TSGN Select the active level of the tool rest arriving signal

0: High level

1: Low level

0 ABOT Weather clear the workpiece coordinate values when power-on

0: Keep the remembered value

1: Clear

004	SANG	HPG	XRC	SOVI	MZRZ	MZRX	ZMZ	ZMX

[Def. value] 00000000

7 SANG Select the function of analog spindle

0: Invalid

1: Valid

6 HPG Select the function of handle control

0: Invalid

1: Valid

5 XRC Programming method of X-axis

0: Diameter programming

1: Radius programming

4 SOVI Select the source of override switch signals

0: From operation panel

1: From external signals. Signals from operation panel are invalid.

3 MZRZ Select the movement direction button controlling Z-axis when returning reference position manually

0: Mask the negative direction button controlling Z-axis

1: Mask the positive direction button controlling Z-axis

2 MZRX Select the movement direction button controlling X-axis when returning reference position manually

0: Mask the negative direction button controlling X-axis

1: Mask the positive direction button controlling X-axis

1 ZMZ Select the direction of Z-axis when returning reference position and perform the backlash compensation at the first time when power-on

0: The negative direction is valid

1: The positive direction is valid

0 ZMX Select the direction of X-axis when returning reference position and perform the backlash compensation at the first time when power-on

0: The negative direction is valid

1: The positive direction is valid

005	Numerator of the electronic gear ratio of X-axis

[Range] 1~127

[Unit]

[Def. value] 1

006	Numerator of the electronic gear ratio of Z-axis

[Range] 1~127

[Unit]

[Def. value] 1

007	Denominator of the electronic gear ratio of X-axis

[Range] 1~127

[Unit]

[Def. value] 10

008	Denominator of the electronic gear ratio of Z-axis

[Range] 1~127

[Unit]

[Def. value] 10

009	Rapid traverse rate of X-axis

[Range] 0~24000

[Unit] mm/min

[Def. value] 4000

010	Rapid traverse rate of Z-axis

[Range] 0~24000

[Unit] mm/min

[Def. value] 6000

011	Specify the backlash compensation distance of X-axis

[Range] 0~2000

[Unit] μm

[Def. value] 0

012	Specify the backlash compensation distance of Z-axis

[Range] 0~2000

[Unit] μm

[Def. value] 0

013	Specify the coordinate value of X-axis after returning the reference position

[Range] -9999999~9999999

[Unit] μm

[Def. value] 0

014	Specify the coordinate value of Z-axis after returning the reference position

[Range] -9999999~9999999

[Unit] μm

[Def. value] 0

015	Stored stoke limitation in the positive direction (X-axis)
016	Stored stoke limitation in the positive direction (Z-axis)
017	Stored stoke limitation in the negative direction (X-axis)
018	Stored stoke limitation in the negative direction (Z-axis)

[Range] -9999999~9999999

[Unit] μm

[Def. value] 9999999 (positive) /-9999999 (negative)

NOTE

1 These parameters are used to specify the maximum distance from the reference position to the travel limitation. The area beyond this range is called inhibition area. In general, set the stored stroke limitations at the maximum length of travel. When machine moves into the inhibition area, an OVERTRAVEL alarm occurs.

2 It takes some time to check weather a machine move into a inhibition area, which means that some allowable travel distance should be considered. Generally, the travel distance equals to about the 0.2 times of the rapid traverse rate. For example, if the rapid traverse rate is 3 m/min, this allowable travel distance is about 0.6 mm (3×0.2).

3 If these parameters are set to 0, the stored stroke limitation function is invalid.

4 If the stored stroke limitation in the positive direction is set to 360000 while the value in the negative direction is set to 0, the corresponding axis works as a rotary axis whose machine coordinate varies from 0.000 to 359.999.

019	LINTX: Time constant of linear acc./dec. control (X-axis)

[Range] 8~4000

[Unit] ms

[Def. value] 100

020	LINTZ: Time constant of linear acc./dec. control (Z-axis)

[Range] 8~4000

[Unit] ms

[Def. value] 100

021	FEEDT: Time constant of exponential acc./dec. control when cutting or feed in HAND mode

[Range] 1~4000

[Unit] ms

[Def. value] 80

022	FEDFL: The lower speed limitation (FL speed)when perform the exponential acc./dec. control

[Range] 0~3000

[Unit] mm/min

[Def. value] 100

023	THRDT: Time constant of exponential acc./dec. control when cutting thread (X-axis)

[Range] 0~4000

[Unit] ms

[Def. value] 100

024	THDFL: Low speed limitation when performing the exponential acc./dec. control in thread cutting (for all axes)

[Range] 6~3000

[Unit] mm/min

[Def. value] 350

025	FEDMX: Upper speed limitation when cutting (effective for all axes)

[Range] 0~20000

[Unit] mm/min

[Def. value] 15000

026	RPDFL: Least speed of the rapid traverse rate (Fo, effective for all axes)

[Range] 6~3000

[Unit] mm/min

[Def. value] 50

027	ZRNFL: Low speed used when returning reference point (effective for all axes)

[Range] 6~3000

[Unit] mm/min

[Def. value] 200

028	JOGFL: Low speed limitation when performing the exponential acc./dec. control in HAND mode (for all axes)

[Range] 0~3000

[Unit] mm/min

[Def. value] 40

029	SEQNIC: Increment of the sequence number when editing a program

[Range] 0~9999

[Unit]

[Def. value] 10

NOTE

When SEQINC is set to 0, the function of inserting a sequence number automatically after inserting EOB is invalid.

030	WLKTME: Filtering time of the input signals

[Range] 0~15

[Unit] 16ms

[Def. value] 1

NOTE

The parameter's default value is 1 and will be checked when power-on. If the value is bigger than 15, it will be automatically set to 1.

031	GRMAX1: The maximum spindle speed of the GEAR 1 when the analog command is 10V.

[Range] 1~9999

[Unit] r/min

[Def. value] 9999

032	GRMAX2: The maximum spindle speed of the GEAR 2 when the analog command is 10V.

[Range] 1~9999

[Unit] r/min

[Def. value] 9999

033	LOWSP: Lower spindle speed limitation when performing the G96 operation

[Range] 0~9999

[Unit] r/min

[Def. value] 99

034	TCTMX: Maximum delay time when performing the tool exchanging operation

[Range] 0~100000

[Unit] ms

[Def. value] 20000

035	MTIME: Delay time when executing an M code

[Range] 1~4080

[Unit] ms

[Def. value] 16

036	CKTDI	QSEL	AGER	QPSL	RVX	RSJG	SSGT	SAGT

[Def. value] 01000000

7 CKTDI Check the state of tool rest in a certain period

0: Don't check

1: Check

6 QSEL Weather the 'Q' address programming is valid or not in thread cutting

0: Invalid

1: Valid

5 AGER Weather the function of automatic changing spindle gear is valid or not

0: Invalid

1: Valid

4 QPSL Select the chuck function

0: Invalid

1: Valid

3 RVX Select the back tool rest function

0: Invalid

1: Valid

2 RSJG Select the operation style of some IO signals when RESET. (M03, M04, M08, M32, M21, M23)

0: Stop outputting these signals

1: Keep the previous output state

1 SSGT Select the GATE PROTECTION function

0: Invalid

1: Valid

0 SAGT Select the working style of GATE PROTECTION function

0: The logic high state of the input signal indicates the gate is closed

1: The logic low state of the input signal indicates the gate is open

037	T1: Delay time from tool rest stops rotating in positive direction to starts rotating in negative direction to clamp

[Range] 1~4080

[Unit] ms

[Def. value] 64

038	TLOCK: Delay time of outputting clamping signal in negative direction

[Range] 1~4080

[Unit] ms

[Def. value] 976

039	TOOLNO: Tool number

[Range] 1~8

[Unit]

[Def. value] 4

040	SPZDTIME: Delay time to brake the spindle

[Range] 1～32640

[Unit] ms

[Def. value] 100

041	QPLS	QPM3	ZG92L	G93N	TWSL	M23O	M21O	LPMH

[Def. value] 01110000

7 QPLS Select the output signal type of chuck

0: Level signal

1: Pulse signal. The pulse width is specified by P51

6 QPM3 Weather check the chuck state when starting the spindle

0: Check the state. If the chuck is loose, system issues an alarm and stops execution.

1: Don't check the state. Ensure that the chuck state is logic high when QPIN is set to 1.

5 ZG92L Select the acc./dec. control style of Z-axis in thread cutting (G92/G76)

0: Exponential acc./dec. control

1: Linear acc./dec. control

4 G93N Select the acc./dec. style when performing the tapping operation (G93)

0: No acc./dec. control

1: Exponential acc./dec. control

3 TWSL Select the tail function

0: Invalid

1: Valid

2 M23O Select the function of BIT5 in DGN.005

0: Not used as output signal for M23

1: Used as output signal for M23

1 M21O Select the function of BIT4 in DGN.005

0: Not used as output signal for M21

1: Used as output signal for M21

0 LPMH Hardware stroke limitation switches' source selection

0: Lower 4-bit in DGN.003 are used as hardware stroke limitation switches

1: Higher 4-bit in DGN.003 are used as hardware stroke limitation switches

042	OFMD2	CHGC	PUCH	SPK2	ESPO	NTHD	G92L	LWN

[Def. value] 10000011

7 OFMD2 1: Select the input style of tool length compensation values

0: No restrictions

1: Only the direct measuring style are valid

6 CHGC Select the background color when equipped with monocolor screens

0: White

1: Black

5 PUCH Select the serial port communication function

0: Invalid

1: Valid

4 SPK2 Select the function of external PAUSE button

0: Invalid

1: Valid

3 ESPO Select the alarm signal's working style

0: Only when EMERGENCY STOP or DRIVER ERROR alarms happen, system outputs alarm signal (M21=0)

1: System outputs alarm signal when any alarm happens (M21=0)

2 NTHD Weather detect the fluctuation of the spindle speed when cutting thread

0: Don't detect

1: Detect. P59 & P60 are valid

1 G92L Select the acc./dec. control style of X-axis in thread cutting (G92/G76)

0: Exponential acc./dec. control

1: Linear acc./dec. control. The time constant used is specified by P57

0 LWN Select the precise thread cutting control

0: Invalid

1: Valid (default value)

043	QPIN	SBDT	JGER	LW	MSTKY	SANG2	AURH	PGRST

[Def. value] 00001000

7 QPIN Weather the chuck state is detected by some input signals

0: No detecting signals

1: Have detecting signals. When starting spindle, these input signals are detected.

6 SBDT Select the optional-block-skip function

0: Invalid

1: Valid

5 JGER Select the function of manually changing analog spindle's gear

0: Invalid

1: Valid. The input signal M42I is used to select gears.

4 LW Select the function of optimized large-pitch-thread cutting control

0: Invalid

1: Valid

3 MSTKY Mask the START button on system panel

0: Mask. The START button on additional panel is valid

1: Not mask

2 SANG2 Select the function of exchanging state between the stepless spindle speed adjusting and the step spindle speed adjusting (It's only available when analog spindle is enabled.)

0: All S commands are treated as stepless spindle speed adjusting commands

1: S1~S4 are treated as step spindle speed adjusting commands. Other S commands are treated as stepless spindle speed adjusting commands.

1 AURH Select the automatic lubrication function

0: Invalid

1: Valid

0 PGRST Weather the cursor returns to the head of the program or not after pressing the RESET button

0: Return to the head

1: Keep the previous state

044	PCMD	M05S	LINT	ZDT1S	SMNREM			

[Def. value] 00010000

7 PCMD Select the command style

0: Maximum speed reaches up to 15 m/min

1: Maximum speed reaches up to 30 m/min

6 M05S Select the work style of M05

0: End the M05 execution after the fixed delay time for normal M codes

1: End the M05 execution after the finishing of brake process.

5 LINT Select the parallel tool rest

0: Invalid

1: Valid

4 ZDT1S Select the work style of the delay time 1 when brake spindle

0: T1 is specified by parameter

1: T1 is fixed as 0 when analog spindle is enabled.

NOTE

When SANG2 is set to 1, the function is invalid. T1 is specified by parameter.

3 SMNREM Weather remembering the number of machined workpiece or not

0: Don't remember

1: Remember

045	STIME1: Delay time 1 for S code when changing gear

[Range] 0～4080

[Unit] ms

[Def. value] 16

046	STIME2: Delay time 2 for S code when changing gear

[Range] 0～4080

[Unit] ms

[Def. value] 16

047	Delay time from spindle stops rotating to outputting brake signal

[Range] 0～4080

[Unit] ms

[Def. value] 500

048	Reserved

[Def. value] 0

049	Reserved

[Def. value] 0

050	Reserved

[Def. value] 0

051	QPLSTIME: Width of chuck controlling pulse signal

[Range] -32768~32767

[Unit] ms

[Def. value] 32

052	Time of automatic lubrication-ON

[Range] 1～65535

[Unit] ms

[Def. value] 16

053	Time of automatic lubrication-OFF

[Range] 1～65535

[Unit] ms

[Def. value] 16

054								
						RHLOW		

[Def. value] 00000000

2 RHLOW Select the function of issuing an alarm when the liquid level of lubrication oil is low

0: Invalid

1: Valid

055	G92XR	XG92P	ZROF					

[Def. value] 00000000

7 G92XR Select the chamfering method in X-axis direction when cutting thread(G92/G76)

0: Chamfering with optional angles (by specifying the J/K commands)

1: When chamfering, X-axis retracts at the rapid traverse rate

6 XG92P Select the retracting method in X-axis direction when cutting thread(G92/G76)

0: Tool retracts with the traditional method

1: Tool retracts in the X-axis direction at the rapid traverse rate

NOTE

1 G92XR is only effective form chamfering. The retracting process (3rd movement in G92/G76 execution) is not affected by G92XR. If the chamfering width is 0, the chamfering action will be omitted, which means that tool retracts directly in the X direction.

2 XG920 is only effective for retracting process and don't affect the chamfering process.

3 When performing the rapid retracting operation, if the chamfering width is 0, only XG92P should be set to 1 (G92XR doesn't work). If the chamfering width is not 0, both XG92P and G92XR should be set to 1.

4 When performing the rapid chamfering operation, it is recommended to use linear acc./dec. control specified by G92L, which will reduce the acc./dec.

5 XG92P=0: Linear acc./dec. time constant in rapid retracting process in X-axis direction is the same as that used by G00 command. XG92P=1: Linear acc./dec. time constant is specified by P58.

6 When G92L is set to 1, the linear acc./dec. time constant of both X-axis and Z-axis when chamfering are the same as that used by G00 command.

5 ZROF Select the rapid traverse rate when returning reference position

0: Using the speed of G00

1: Specified by P41' and P42'

056	BAUTE: Baud rate of RS232 port

[Range] 2400, 4800

[Unit] bps

[Def. value] 2400

057	G92LINTX: Linear acc./dec. time constant when controlling X-axis in thread cutting (G92L=1)

[Range] -32768~32767

[Unit] ms

[Def. value] 80

058	G92LINTZ: Linear acc./dec. time constant when controlling Z-axis in thread cutting (G92L=1)

[Range] -32768~32767

[Unit] ms

[Def. value] 80

059	Circle number used when calculate the average speed of the spindle when cutting thread

[Range] -32768~32767

[Unit] circle

[Def. value] 4

060	Allowable fluctuation of the spindle speed when cutting thread

[Range] -32768~32767

[Unit] %

[Def. value] 5

NOTE

P59 & P60 are valid when NTHD is set to 1. Setting P60 to 5 means that only when the fluctuation of spindle speed locates in 5%, thread cutting will be performed.

1.2 Compound Parameters

Compound parameters will be displayed by pressing [SHIFT] in the 2nd and 3rd page. Compound parameters and the original ones use the same parameter number. This section uses the following method to describe these compound parameters.

- P21——Represents the original parameter
- P21'——Represents the compound parameter corresponding to P21

Meanings of compound parameters are described below:

021'	MRCCD: Cutting depth of multiple cycles (G71, G72)

[Range] -99999999~99999999

[Unit] mm

[Def. value] 0

022'	MRCDT: Retracting amount of multiple cycles (G71, G72)

[Range] -99999999~99999999

[Unit] μm

[Def. value] 500

023’	PECSCX: Total cutting amount in X-axis direction for multiple cycles (G73)

[Range] -99999999~99999999
[Unit] μm
[Def. value] 1000

024’	PECSCZ: Total cutting amount in Z-axis direction for multiple cycles (G73)

[Range] -99999999~99999999
[Unit] μm
[Def. value] 2000

025’	PATIM: Cutting times of multiple cycles (G73)

[Range] -99999999~99999999
[Unit] time
[Def. value] 2

026’	GROVE: Retracting amount for multiple cycles (G74, G75)

[Range] -99999999~99999999
[Unit] μm
[Def. value] 500

027’	THRPT: Repetition times of finishing cutting for multiple cycles (G74, G75)

[Range] -99999999~99999999
[Unit] time
[Def. value] 2

028’	THDCH: Chamfering width in thread cutting (G92, G76)

[Range] -99999999~99999999
[Unit] 0.1 × pitch
[Def. value] 0

029’	THANG: Tool nose angle for thread cutting (G76, G78)

[Range] 0, 29, 30, 55, 60, 80
[Unit] degree
[Def. value] 0

030’	THCLM: Least cutting depth in thread cutting (G76, G78)

[Range] -99999999~99999999
[Unit] μm
[Def. value] 20

031’	THDFN: Amount for finishing cutting (G76)

[Range] -99999999~99999999
[Unit] mm
[Def. value] 20

032’	Feed direction selection for each cutting used by G78

[Range] -99999999~99999999
[Unit]
[Def. value] 0

033’	Feed method for each cutting used by G78

[Range] -99999999~99999999

[Unit]

[Def. value] 0

034’	Speed for screw-in and chamfering used by G78

[Range] -99999999~99999999

[Unit] mm/min

[Def. value] 2500

035’	Date: year
036’	Date: month
037’	Date: day
038’	Time: hour
039’	Time: minute
040’	Time: second

041’	ZRNRPD1: Rapid traverse rate when returning reference position (X-axis)

[Range] -32768~32767

[Unit] mm/min

[Def. value] 4000

042’	ZRNRPD2: Rapid traverse rate when returning reference position (X-axis)

[Range] -32768~32767

[Unit] mm/min

[Def. value] 4000

043’	Reserved

[Def. value] 0

044’	Reserved

[Def. value] 0

1.3 Parameter List

No.	Def. value	Description
1	01001100	Bit parameters
2	00000000	Bit parameters
3	00001110	Bit parameters
4	00000000	Bit parameters
5	1	Numerator of the electronic gear ratio of X-axis
6	1	Numerator of the electronic gear ratio of Z-axis
7	10	Denominator of the electronic gear ratio of X-axis
8	10	Denominator of the electronic gear ratio of Z-axis
9	4000	Rapid traverse rate of X-axis
10	6000	Rapid traverse rate of Z-axis
11	0	Specify the backlash compensation distance of X-axis

12	0	Specify the backlash compensation distance of Z-axis
13	0	Specify the coordinate value of X-axis after returning the reference position
14	0	Specify the coordinate value of Z-axis after returning the reference position
15	9999999	Stored stoke limitation in the positive direction (X-axis)
16	9999999	Stored stoke limitation in the positive direction (Z-axis)
17	-9999999	Stored stoke limitation in the negative direction (X-axis)
18	-9999999	Stored stoke limitation in the negative direction (Z-axis)
19	100	LINTX: Time constant of linear acc./dec. control (X-axis)
20	100	LINTX: Time constant of linear acc./dec. control (Z-axis)
21	80	Time constant of exponential acc./dec. control in HAND mode
22	100	The lower speed limitation when perform the exponential acc./dec. control
23	100	Time constant of exponential acc./dec. control when cutting thread (X-axis)
24	350	Low speed limitation of exponential acc./dec. control in thread cutting
25	15000	FEDMX: Upper speed limitation when cutting (effective for all axes)
26	50	RPDFL: Least speed of the rapid traverse rate (Fo, effective for all axes)
27	200	ZRNFL: Low speed used when returning reference point (effective for all axes)
28	40	JOGFL: Low speed limitation when performing the exponential acc./dec. control in HAND mode (for all axes)
29	10	SEQNIC: Increment of the sequence number when editing a program
30	1	WLKTME: Filtering time of the input signals
31	9999	The maximum spindle speed of the GEAR 1 when the analog command is 10V.
32	9999	The maximum spindle speed of the GEAR 2 when the analog command is 10V.
33	99	LOWSP: Lower spindle speed limitation when performing the G96 operation
34	20000	TCTMX: Maximum delay time when performing the tool exchanging operation
35	16	MTIME: Delay time when executing an M code
36	01000000	Bit parameter
37	64	T1: Delay time from tool rest stops rotating in positive direction to starts rotating in negative direction to clamp
38	976	TLOCK: Delay time of outputting clamping signal in negative direction
39	4	TOOLNO: Tool number
40	100	SPZDTIME: Delay time to brake the spindle
41	01110000	Bit parameter
42	10000011	Bit parameter
43	00001000	Bit parameter
44	00010000	Bit parameter
45	16	STIME1: Delay time 1 for S code when changing gear
46	16	STIME2: Delay time 2 for S code when changing gear
47	500	Delay time from spindle stops rotating to outputting brake signal
48	0	Reserved
49	0	Reserved
50	0	Reserved

51	32	QPLSTIME: Width of chuck controlling pulse signal
52	16	Time of automatic lubrication-ON
53	16	Time of automatic lubrication-OFF
54	00000000	Bit parameter
55	00000000	Bit parameter
56	2400	BAUTE: Baud rate of RS232 port
57	80	Linear acc./dec. time constant of X-axis in thread cutting (G92L=1)
58	80	Linear acc./dec. time constant of Z-axis in thread cutting (G92L=1)
59	4	Circle number used when calculate the average speed of the spindle when cutting thread
60	5	Allowable fluctuation of the spindle speed when cutting thread

1.4 Descriptions of Parameter Setting

1.4.1 Emergency stop switch, pause, increment step selection, soft/hard limit setting

Emergency stop switch 1

It can be masked by MESP.

001			MESP					

5 MESP Mask the signal *ESP1

0: Not mask

1: Mask

System panel switch signal (emergency stop 2, feed hold, spindle pause)

It can be masked by MSPL.

001	MSPL							

7 MSPL Mask the signals from panel including *ESP2, *SPL and *SPK

0: Not mask

1: Mask

Stored stroke limitation checking

When machine moves out of the available range specified by P15~P18, system issues an alarm and stop moving any axis. In manual mode, move corresponding axis in the reverse direction to exit from forbidden area, and press [RESET] to clear the alarm. Set both the positive and negative limitations to 0 will disable the stored stroke limitation checking function.

Hardware stroke limitation checking

Use MOT to control the function.

001		MOT						

6 MOT Mask the hardware travel limitation switches

0: Not mask

1: Mask

041								LPMH

0 LPMH Hardware stroke limitation switches' source selection

0: Lower 4-bit in DGN.003 are used as hardware stroke limitation switches

1: Higher 4-bit in DGN.003 are used as hardware stroke limitation switches

Step/handwheel increment selection

SINC is used to disable the 0.1 and 1 increment selections.

001				SINC				

4 SINC Mask the incremental steps 0.1 and 1 used in HANDLE and SINGLE-STEP mode

0: Not mask

1: Mask

NOTE

It is recommended to set SINC to 1 when equipped with step motors.

1.4.2 Parameters setting for reference position returning

System supports several methods for returning reference position. Associate parameters are described as follows

1.4.2.1 Bit parameters

MZRZ/MZRX

Select the valid direction keys for each axis when perform the reference position returning operation.

ZMX/ZMZ

Select the direction of returning reference position. They are valid only when mechanical reference position switches are used.

When the two directions specified by MZRn and ZMn (n represents X or Z) are the same, system returns reference position along the direction specified until reaching the reference position.

When the two directions specified by MZRn and ZMn (n represents X or Z) are reverse, system returns reference position in a complex process as described in below diagram.

ZNIK

Weather the direction keys when returning reference position is self-hold or not.

NOTE

For safety consideration, it is recommended to set ZNIK to 0. Setting ZNIK to 1 is more convenient for operation.

ZRSZ/ZRSX

Select weather having mechanical reference position or not

ZRSn=1: Have mechanical reference position. Axis moves toward the reference position at a rapid traverse rate specified until press down the deceleration switch, and continue moves at a low speed to reach the reference position.

ZRSn=0: No mechanical reference position. When performing reference position retuning operation in this case, system moves axis to the point whose machine coordinate value is 0. Method for setting floating reference position is as follows:

In POS interface, press [CAN] and X at the same time, the machine coordinate value of X-axis is cleared to 0. Perform the same operations to set Z-axis.

ZCX/ZCZ

ZCn=0: Deceleration switch and zero-point signal are needed

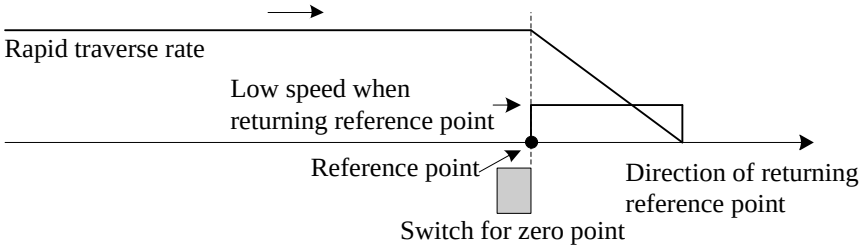
ZCn=1: Type C reference position retuning

NOTE

1 Only when there is a mechanical zero, the parameter has meaning.

2 When a step motor having no one revolution signal is used, and only one proximity switch is used for convenience in installation, set this parameter. When a servo motor having one revolution signal is used, don't set this parameter.

3 You must set MZR* and ZM* to be inconsistent. In the figure above, MZR* is set to 0 (+ direction key is valid), ZM* is set to 1 (- direction homing)



1.4.2.2 Data parameters

ZRNFL

ZRNFL is used to specify the low speed when returning to reference position.

When there is a mechanical zero, the parameter is valid. At homing, the axis will move at this low speed when comes across the deceleration switch. The lower this speed is, the higher accuracy in homing. But a too low speed will influence efficiency.

RPDFX/RPDFZ

The two parameters are used to specify the rapid traverse rate used in the first stage of returning reference position.

1.4.3 Parameter setting for analog spindle

When using an analog spindle, set the following parameters correctly.

SANG

SANG is used to enable analog spindle.

Command like S+4-digit are available when setting SANG to 1. The rotation speed corresponding to a certain S code should be adjusted by setting parameters.

GRMAX

The spindle speed when outputting analog voltage is 10V. Two method if available for setting GRMAX:

- **Method 1**

According to spindle motor's speed and the gear ratio of spindle to spindle motor when outputting 10V analog voltage, calculate the value of GRMAX.

- **Method 2**

Using the factory-set parameters, specify S9999 command and measure the rotation speed of spindle. Set GRMAX to the measured value.

LOWSP

It is used to specify the lower spindle speed limitation when performing the G96 operation.

In constant cutting speed mode, spindle speed varies according to the coordinate of X-axis. In this case, the spindle speed may be too low which is improper for cutting. To avoid this case happening, use LOWSP to set a lowest spindle speed in constant cutting speed mode. When the calculated spindle speed reaches the value specified by LOWSP, system rotates at the fixed lowest speed.

1.4.4 Parameter setting for backlash compensation

CPF4,3,2,1

Select the pulse frequency when perform backlash compensation operation.

Compensation frequency = (set value+1) Kpps.

For servo driver: CPF*＝ 1100.

For step driver: CPF* ＝0000.

BDEC

Select the method for output pulses.

BDED=0: Output pulses in a fixed frequency (CPF4,3,2,1 and BD8 setting).

BDED=1: Output pulses in an acceleration/deceleration mode, parameter

For servo driver, set BDEC to 0 to achieve better machining effect.

For step driver, set BDEC to 1 to avoid losing steps.

BD8

BD8=0: Backlash compensation is performed at the frequency set by P001

BD8=1: Backlash compensation is performed at 1/8 of the frequency set by P001

For servo driver, set BD8 to 0.

For step driver, set BD8 to 1.

RVDL

RCDL=0: When changing movement directions, direction signal and pulse signal are output at the same time.

RCDL=1: When changing movement directions, output direction signal firstly, and output pulse signal after a certain delay time.

For servo driver: set 0.

For step driver: set 1.

KSGN

0 : axis - direction movement, movement sign does not remain.

1 : axis - direction movement, movement sign remains.

When single direction pulse output is selected: set 1. When bidirectional pulse output is selected: the parameter is meaningless.

DALZ/DIRX (parameter P002 bit3 2): selecting alarm voltages for the driver

This parameter can be set if different driver are equipped, because different driver have different alarm voltages.

DIRZ/DIRX (P002 bit1 0): selecting motor rotation direction

Changing the parameter may change motor rotation direction.

1.4.5 Parameter setting for thread cutting

When cutting thread, at start and end part for cutting, as a result of acceleration/deceleration, points with incorrect lead will occur. This can be solved by such a way that the commanded thread length is longer than the needed thread length. If the commanded thread length is limited, especially at the end part, you can set that by adjusting several parameters relative with thread cutting.

P021

It is time constant of exponential acc./dec. control when cutting or feed in HAND mode.

1 Reducing the parameter can shorten acceleration /deceleration time at thread cutting, which makes the machined thread leads be consistent.

2 For a step driver: range: 50~100. Providing that the step motor does not miss step, reduce the value as far as possible.

3 For a servo driver: range: 40~80. Providing that the servo motor does not generate oscillation, reduce the value as far as possible.

P022

It is the lower speed limitation (FL speed)when perform the exponential acc./dec. control Low limit of low speed (FL speed) for exponential acceleration/deceleration for cutting feed and manual feed

Range:100~300

P023

It is the time constant of exponential acc./dec. control when cutting thread (X-axis)

The setting of this parameter can be of the same value as P021 parameter.

P024

It is the lower speed limitation when performing the exponential acc./dec. control in thread cutting (for all axes).

P025

It is the upper speed limitation when cutting (effective for all axes).

When cutting a big pitch thread, if thread pitch × spindle speed is bigger than the value set by this parameter, it is impossible to machine thread with correct pitch. Therefore, the setting of this parameter must be increased to machine big pitch thread. The maximal cutting feedrate of K100Ti-B can be up to 8000 mm/min, usually it is set as 6000 mm/min.

P'028

It is the chamfering width in thread cutting (G92, G76).

At second Parameter page, pressing 

to display another meaning of P'028 parameter, which is chamfering width at thread cutting (G92, G76), that is, retreat amount for thread. The default setting of P'028 parameter is 10 which means the retreat width is 1 pitch. At thread cutting, the retreat amount shall be reduced as far as possible. The minimal setting can be 1, and retract is conducted in advance from the point that is 0.1 pitch from the thread end. In addition, the P retreat in G92 block will also change PA28 value, and it will keep invariable after power OFF. Generally, for thread retreating without special requirements, you may not specify P retract in the program, instead you can adopt directly the P'028 setting, P'028=1.

In addition, when using G92/G76 to cut thread, you may select to adopt linear acceleration/deceleration and let X axis retreats at G00 traverse rate. For the setting and use method, refer to chapter II.

1.4.6 Step and Handwheel

If the system is equipped with a handwheel, set the following parameter:

Set HPG (P004 bit6) to 1 to enable handwheel function, while clear it to 0 to select step mode.

1.4.7 Saving electronic disk

Please save after adjusting a program. When the system runs, the data that the battery held are used. Strong external interfere can confuse memory data when power ON or OFF. Reading disk at power ON not only can restore data but also restore the messy memory.

1.4.8 Setting electronic gear ratio

When lead screws of different pitches are matched with motors of different step angles or servo motors with different one-revolution pulses, or are connected via different speed-varying gears, it is possible to keep consistent the programmed and the actual movement distances by setting the system's electronic gear ratio.

Step Motor

$$\frac{CMR}{CMD} = \frac{360}{a \times L \times 1000 \times C}$$

CMR─ command multiplication ratio (parameter №005～006)

CMD─ command frequency division coefficient (parameter №007～008)

a─ step angle (degree)

L─ machine tool movement corresponding one revolution of step motor (mm)

C─ normally set to 1; when X axis is diameter programming, set to 2.

Example a = 0.75 L = 5

$$\frac{CMR}{CMD} = \frac{12}{125}$$

The system's minimum output unit is CMD/CMR =125/12 (unit: 0.001 mm)

NOTE

1 For motors of any step angles, the system's minimum programming unit is 0.001 mm, and the minimum output unit is depending upon a and L. The lower a and L are, the higher resolution will be, but the speed will be low. On the contrary, the higher a and L are, the higher the speed will be, but the lower the resolution will be.

2 Range:1~127

Servo Motor

$$\frac{CMR}{CMD} = \frac{P}{L \times 1000 \times c}$$

L─ machine tool movement corresponding to servo motor's one revolution (mm)

P─ pulse number corresponding to motor's one revolution feedback

1.4.9 Setting acc./dec. time constant

When a step motor is used, in order to avoid rotation blocking, the exponential or linear acceleration /deceleration time constant is greater than that with a servo motor. Set time constant depending upon actualities.

G00 linear acceleration/deceleration time constant: 200~500 (P019, 020)

G01 exponential acceleration/deceleration time constant: 50~100 (P021)

If the exponential acceleration/deceleration time constant is too long, set parameter P022.

1.4.10 Setting rotary axis

X/Z axis can be specified as rotary axis.

For an axis, when its positive limit parameter is set as 360000, negative limit parameter as 0, the axis is specified rotary axis.

Example

If parameter 015=360000 and parameter 017=0, X axis is rotary axis.

When the axis is specified as an rotary axis, the display range for machine tool coordinate is: 0~359.999

2 DIAGNOSIS LIST

2.1 Standard Diagnosis Signals

2.1.1 Input signals

000			*DECX	QPI				
001			*DECZ	*ESP1	T04	T03	T02	T01
002					*ESP2	*SPL	*SPK	ST
003	X37	X36	X35	X34	X33	X32	X31	X30

2.1.2 Output signals

004	SPZD	TL-	TL+	M32	M08	STL	M04	M03
005	QPJ	QPS	Y25	ESPO	S4	S3	S2	S1

2.1.3 Status signals

006	TMLT	CSCT	CITL	COVZ	CINP	CDWL	CMTN	CFIN
007	STP	REST	EMS		RSTB			CSU

2.1.4 MDI key board signals

008	W	U/6	Z/5	X/4	R/3	F/2	H/1	S/0
009	K	J	I	P	T/.	M/-	G/9	N/8
010	O/7	L	CRL	CRR	PGU	PGD	CRU	CRD
011	GRPH SHIFT	TST STR	ALM EOB	DGN OUTP	PRM INP	OFS DEL	PRG INS	POS ALT
012	HDL+ RESET	HDL- CAN	START SLH	RPD RSV	Z- BRT	Z+ OPGH	X- BRT-	X+ BRT+
013	RPD+ FD+	RPD- FD-	EDIT CHTL	AUTO COOL	MDI JOG	MANU STOP	RZN REV	STEP FWD

014	RERRX: Track error/Output pulse number (X-axis)
015	RERRZ: Track error/Output pulse number (Z-axis)

NOTE

Press [SHIFT] to change the contents of displaying.

2.2 Additional Diagnosis Signals

Press [INS] and [1] to enter the interface displaying additional diagnosis signals. Press [CAN] to return to the interface displaying standard diagnosis signals.

Diagnosis signals in DGN.0~DGN.3 are the same with those in standard interface.

2.2.1 System interface signals

006			RFZ	RFX		PCS	PCZ	PCX
007							ALMZ	ALMX

2.2.2 Input signals to CNC

008	HX/RV1		*DECX		-X	+X		
009	HZ/RV2		*DECZ		-Z	+Z		
010	DRN				GR2	GR1		
011	MLK	MP2	MP1		SBK	BDT		
012	ZRN	*SSTP	SOR	SAR	FIN	ST	STLK	MIR1
013	ERS	RT	*SP	*ESP	*OV8	*OV4	*OV2	*OV1
014	PN8	PN4	PN2	PN1	KEY	MD4	MD2	MD1
015	CDZ	SMZ	AFL	OVC		SOVC	SOVB	SOVA

3 ALARM LIST

3.1 Program Operation Alarms (P/S Alarm)

Number	Content
000	Turn off then on again the power after the parameter was input
003	Data exceeding the maximum allowable number of digits was input (See the section on max. programmable dimensions).
004	A number, a sign (-) or a decimal point was input without an address at the beginning of a block.
005	The address was not followed by proper data but was followed by another address or EOB code.
006	Sign "-" input error. (Sign"-" was input after an address with which it can't be used. Or two or more "-" signs were input.)
007	Decimal point "." input error. (A decimal point was input after an address with which it can't be used. Or two or more decimal points were input.)
009	An invalid character was input in the significant information zone
010	An invalid G code was specified.
011	The federate was not specified for cutting feed or the federate was inadequate.
023	A negative value is specified as the radius of circular interpolation.
029	An offset value with T code is too large.
030	An offset number with T code is too large.
060	The specified sequence number was not found in the sequence number searching.
061	The address P or Q is not specified in command G70/G71/G72/G73 .
062	(1) Depth of cut for multiple cycles (G71,G72) is zero or negative. (2) The repetitive count for multiple cycles (G73) is zero or negative. (3) The value of ΔI or Δj for G74 or G75 is negative. (4) The value of ΔI or Δj for G74 or G75 is not zero, but the value specified by U or W is not lager than zero. (5) The retraction direction is specified but the value of Δd is negative. (6) The depth of cut for G76 is negative. (7) The minimum depth of cut is larger than the total depth of cut for G76. (8) Angle of tool nose for G76 is invalid.
063	The sequence number specified in G70/G71/G72/G73 command cannot be found.
065	(1)The program specified by address P for G71/G72/G73 cycle contains no G00 or G01. (2)The program specified by address P for G71 cycle contains address Z (W). or The program specified by address P for G72 cycle contains address X (U).
066	The program specified by address P and Q for G70/G71/G72 contains invalid G code.
067	In MDI mode, the command G70/G71/G72/G73 with P or Q is entered.

Number	Content
068	The memory is full.
071	The searched address was not found. Or the specified program was not found.
072	The number of program exceeds 63.
073	The program number has already used.
074	The program number is other than 1 to 9999.
076	Address P was not specified in a M98 block.
077	The subprogram exceeds the limit for nesting depth.
078	The program number or the sequence number which was specified by address P in a M98 block was not found between M98 and M99.
090	An attempt is made to perform reference point returning before setting zero.
101	The power is OFF while rewriting the contents of memory/program. Power ON again after power OFF can automatically cancel the alarm.
148	Macro error: cannot find the specified sequence number

3.2 Overtravel Alarms

Number	Contents
+X	Over +X travel limit
-X	Over -X travel limit
+Z	Over +Z travel limit
-Z	Over -Z travel limit

3.3 Driver Alarm

Number	Contents	Remarks
12	Driver alarm for X-axis	driver malfunction or connection cable malfunction.
22	Driver alarm for Z-axis	
13	Federate is too large for X axis. It is generated by incorrect setting of CMR or CMD	The alarm is usually due to CMR or CMD setting error, or because the commanded speed is over maximum.
23	Federate is too large for Z axis. It is generated by incorrect setting of CMR or CMD	

3.4 CNC Alarm

Number	Contents
02	CMOS error
03	ROM parity alarm
06	WATCH DOG alarm
07	CPU error (0,3,4,6 alarm)
08	Illegal non-masking interruption

3.5 External Message Alarm

Number	Contents	Remarks
01	M code alarm. Invalid M code was specified in program.	
02	S code alarm. Invalid S code was specified in program.	
03	T code alarm. Invalid T code was specified in program.	
05	Delay time for tool selection is over upper limit.	
06	M04 was incorrectly specified. While Spindle is rotating CW (CCW), M04 (M03) was commanded without stopping the spindle.	
08	The parameter of total tool number is wrong.	
12	Start key/switch is not released	It takes more than 1 s to close
13	Start key/switch jumps abnormally	Closing over 3 times in 1s
15	Spindle is started with unclamped chuck	