CNC Series

KND—1TBⅡ、1TBⅢ、K2T CNC FOR LATHES

USER'S MANUAL

BeiJing KND CNC Technical Co.,Ltd

B08/10B-T00N-0003A

CONTENTS

Ⅰ GENERAL

KND－1TBⅡ a economic CNC to control digital servo motor or stepper motor for lathe or common two axis mechanism has been developed by Beijing KND CNC technique Co. Ltd. to satisfy the NC markets with high performance and low cost. The high-speed microprocessors, custom LSIs, multiple-layer PCBs and high resolving power LCD are adopted to simplify the structure without functions losing . The CNC system is more compact and high reliability and flexility is available . The additional panel and the CNC operator's panel is integrated which is very helpful to the machine builder. The operator's panel is very friendly with fluent online help.So the operation is easy. KND－1TBⅡ provides high-performance and significantly improves the performance / cost ratio.

This manual explains programming, operation, and connection about KND－1TBⅡ CNC system.

All functions available in KND－1TBⅡ CNC are described in this manual. Refer to the machine builder's manual for actual functions available .

The document contained by KND－1TBⅡ CNC system as follows：

KND－1TBⅡ user's manual

Type in K1T series：

- K1TB ： 320×240 Graph LCD，Panel dimension 400X170mm
- K1TB Ⅱ ： 320×240 Graph LCD，Panel dimension 400X245mm
- K1TB Ⅲ ： 320×240 Graph LCD，Panel dimension 320X245mm
- K1TA ： 192×64 Graph LCD， Panel dimension 160X245mm
- K1TA Ⅱ ： 192×64 Graph LCD， Panel dimension 400X245mm

The function and operation of deffferen types are not same.

This manual is for K1TBⅡ,K1TBⅢ and K2TAB with the main board version 08I-W01Z-0108（V8）and the software version is K1TBA01 031220. If other vertion is adopted, the defference is described in "Additional manual".

Ⅱ PROGRAMMING

The NC machine tool machines a part automaticaly by using the part program . With a good part program the machine tool can machine the part according to the drawing,and furthermore, it can work more efficiently .

The program used by KND CNC is compatible with FANUC.

In this section the following functions will be described: Format used to program functions in the NC language, characteristics, and restrictions. Before a program is created, refer to this section carefully.

1 Coordinate system

There are two contolled axis , X and Z .The two axes X and Z can be controlled simutaneously ,so the tool can move along straight lines and arcs. Each contrlled axis is specified by the address of the demension word used by the NC. In the absolute command, the address word （X, Z ）and the coordinate value of the end point is programmed; in the incremental command, the address word （U,W）and the move distance of the axis itself is programmed .These address words may be specified in the same block as (X,W) or (U,Z). Dimensions of the X axis is set in diameter in this CNC system .

1.1 Absolute commands

The tool moves to a point at ”the distance from zero point of the coordinate system” that is to the position of the coordinate values.

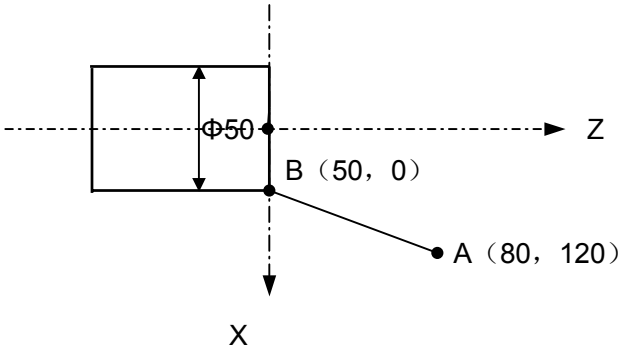


Figure 1-1 Absolute commands and Incremental comands

In absolute commands, a command to move a tool from A to B（see Figure 1-1) at the rapid traverse rate is as follows: G00 X50 Z0；

1.2 Incremental comands

Specify the distance from the previous tool position to the next tool position.

In Incremental commands, a command to move a tool from A to B（see Figure 1-1) at the rapid traverse rate is as follows: G00 X50 Z0；

1.3 Absolute and Incremental comands in the same block

For convenience，Absolute comands and Incremental comands may be specified in the same block as (X,W) or (U,Z). But (X,W) or (U,Z) should not be used together in the same block.

1.4 Least input increment

The least input increment for both axis is 0.001mm. But the least command increment for X axis is 0.0005mm，for Z axis it is 0.001mm .

2 Program Components

When machining a part using an NC machine tool, the tool path and other machining conditions are programmed.This is called a program.

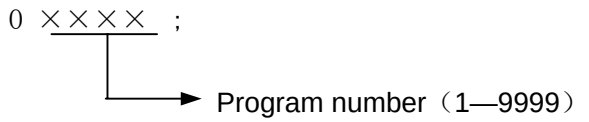
A program is composed of several blocks. A block is composed of one or more words. One block is sepreated from another block with the program block end code‘；’.

2.1 Program general 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 the Program start code and Program end code are both ‘%’ defined in the ISO Standard .When a program is writed, the Program end code and Program start code is added automatically. The Program end code is always displayed but the Program start code is never.

2.1.1 Program number

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



2.1.2 Program command section

Every program comprises many commands. Its command unit is named program block. One block is sepreated from another block with the program block end code’ ; ’. A block contains information necessary for machining, such as a move command or coolant on/off command. At the head of a block, a sequence number consisting of address N followed by a number not longer than five digits (1 to 9999) can be placed.

General program block format：

N××××	G××	X×××Z×××	M××	S××	F××	T××	；
Sequence number	Preparatory function	Dimension word	Miscellaneous function	Spindle speed function	Feed function	Tool function	program block end code

(1) Sequence number

Sequence numbers can be specified in a random order, and any numbers can be skipped. Sequence numbers may be specified for all blocks or only for desired blocks of the program. In general, however, it is convenient to assign sequence numbers in ascending order in phase with themachining steps

Sequence number automatic increment function：

When parameter P029 is not specified 0, sequence number automatic increment function is valid. When pressing EOB to end compiling the current, the sequence number of the next program block will give birth to automatically. The parameter value specified by P029 is the relative increment. When a new sequence number is inserted, the following sequence number will increase from the inserted number .

(2) Word and address

A block consists of one or more words. A word consists of an address followed by a number some digits long. (The plus sign (+) or minus 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. Following table indicates the usable addresses and their meanings.

The same address may have different meanings, depending on the preparatory function specification.

Function	Address	Meaning
Program number	O	Program number
Sequence number	N	Sequence number
Preparatory functions	G	Specify motion status(linear, arc, etc.)
Dimension word	X, Z, U, W	Coordinate axes motion command
	R	Arc radius
	I, K	Position of arc center, chamfer amount
Feed function	F	Rate of feed per minute, Rate of feed per revolution
Spindle speed function	S	Spindle speed
Tool function	T	Tool number designation
Auxiliary functions	M	On/off control on the machine tool
Dwell	P, U, X	Dwell time
Program number designation	P	Subprogram number
Number of repetitions	P	Number of subprogram repetitions

(3) Program end

The following codes indicate the end of a program at the end of the program.

For main program, M30 indicates the end of the main program. If it is executed in program execution, the CNC terminates the execution of the program, and the reset state is set. The main program may be excuted from start by pressing ‘CYCLE’.

M99 indicates the end of a subprogram. When the subprogram end code is executed, control returns to the program that called the subprogram.

2.2 Main program and subprogram

(1) Main program

There are two program types, main program and subprogram. Normally, the CNC operates according to the main program. However, when a command calling a subprogram is encountered in the main program, control is passed to the subprogram. When a command specifying a return to the main program is encountered in a subprogram, control is returned to the main program.

The CNC memory can hold up to 63 main programs and subprograms. A main program can be selected from the stored main programs to operate the machine.

(2) Subprogram

If a program contains a fixed sequence or frequently repeated pattern, such a sequence or pattern can be stored as a subprogram in memory to simplify the program. A subprogram can be called from the main program. A called subprogram 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.

Subprogram call format:

M98　××　××××;

The called subprogram number must be 4-digitand the leading zero must not be omited. Range（0001～9999）

Number of times the subprogram is called repeatedly（1～999）

When no repetition data is specified, the subprogram is called just once.

Examples：

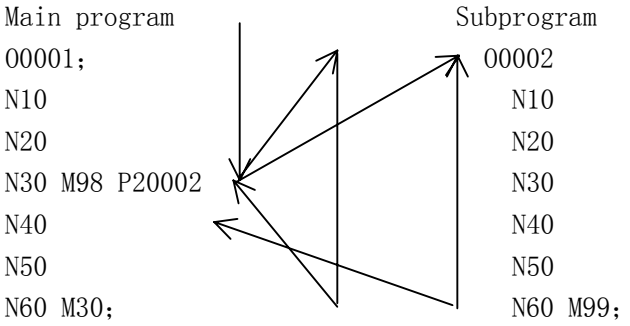
M98 P51002；

This command specifies "Call the subprogram (number 1002) five times in succession." A subprogram call command (M98P_) can be specified in the same block as a move command.

X1000 M98 P1200；

This example calls the subprogram (number 1200) after an X movement.

Execution sequence of subprograms called from a main program：



A subprogram can call another subprogram in the same way as a main program calls a subprogram.

Special Usage

If M99 is executed in a main program, control returns to the start of the main program, then execution is repeated starting at the head of the main program. The execution will not stop untill the RESET key is pressed.

Note:

1：If the subprogram number specified by address P cannot be found, an alarm (No. 078) is output.

2：I the MDI mode, M98 P×××× is locked.

3 Program code

This chapter describes all the program code that are supported by the KND1TBⅡ CNC.

3.1 G function— Preparatory functions

A number following address G determines the meaning of the command for the concerned block. G codes are divided into the following two types..

1) One-off G code: Effectivet in the block specified
2) Modal G code: Effectivet before another G code in the same group

(example) G01 and G00 are modal G code in the same group:

G01 X_F_； X axis feed as F speed
Z_； Z axis feed as F speed, G01 is effective
G00 Z_； G01 is uneffectual, G00 is effective

Preparatory function meanings table

G code	group	function
G00	01	Positioning (Rapid traverse)
*G01		Linear interpolation(cutting feed)
G02		Circular interpolation CW
G03		Circular interpolation CCW
G04	00	Dwell
G28	00	Return to reference position
G32	01	Thread cutting
G50	00	Coordinate system setting
G70	00	Finishing cycle
G71		Stock removal in turning
G72		Stock removal in facing
G73		Pattern repeating
G74		End face peck drilling
G75		Outer diameter/internal diameter grooving
G76		Multiple threading cycle
G90	01	Outer diameter/internal diameter cutting cycle
G92		Thread cutting cycle
G93		Fixed threading cycle
G94		Endface cutting cycle
G96	02	Constant surface speed control
*G97		Constant surface speed control cancel
*G98	03	Per minute feed
G99		Per revolution feed

Note 1: G codes marked * are enabled when the power is turned on or the CNC is reset. For example,G98 is enabled without executing G98 command .

Note 2: The G code of group 00 are not modal. They are effective only in the block in which they

are specified.

Note 3: If a G code not listed in the above table is specified, an alarm (No. 010) is output..

Note 4: A number of G codes can be specified in a block . When more than one G code of the same group is specified, the G code specified last is effective.

3.2 Basic preparatory functions

Basic preparatory functions include G00,G01,G02,G03,G28,G32,G50 and so on.

(1) Positioning(G00)

The G00 command moves a tool to the position in the workpiece system specified with an absolute or an incremental command at a rapid traverse rate.

In the absolute command, coordinate value of the end point is programmed.

In the incremental command the distance the tool moves is programmed.

Format: G00 X（U）_ Z（W）;

X（U）_ Z（W）are specified coordinate value

Example of G00(see Figure 3-1)

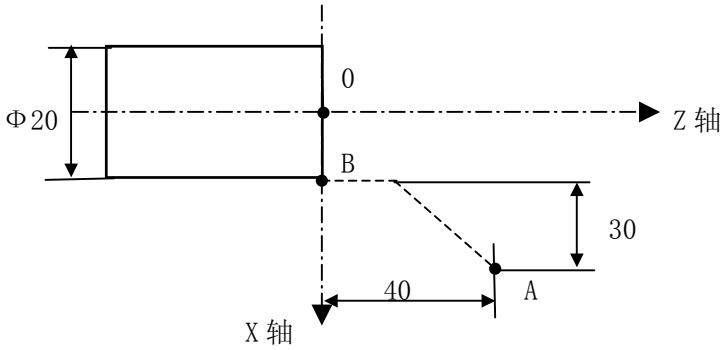


Figure 3-1 Positioning

Diameter programmimg . The tool moves from A to B at the rapid traverse rate.

G00 X20 Z0; (Absolute command)

or

G00 U－60 W－40; (Incremental command)

Note 1: The tool move to the specified position for each axis independently. Each axis moves at the predefined rapid traverse rate independently .

Note 2: The rapid traverse rate in the G00 command is set for each axis independently by the machine tool builder. Accordingly, the rapid traverse rate cannot be specified in the address F.

Fourstep rapid traverse override is effective during rapid traverse, 100%,50%,25% and F0. The rapid traverse override can be adjusted by pressing the rapid traverse override key .The effective rapid traverse override is displayed on the down left corner of the position page .

Note 3: G00 is modal command,When the following command is also G00, it can be omitted. G0 is equivalent to G00.

Note 4: Pay special attention to the position of the tool when X,Z axes rapidly traverse at the same time to avoid colliding.

(2) Linear Interpolation G01

Format: G01 X(U)_ Z(W)__ F__ ;

The G01 command moves a tool along a line to the (X, Z) position in the work coordinate system or from it's current position to the position specified by the (U, W) values at the feedrate specified in F. The feed distance is specified by IP. The feedrate specified in F is effective until a new value is specified. It need not be specified for each block.

Example: See Figure 3－2

Diameter programmimg . The tool moves from A to B along a line at the feedrate specified in address F.

G01 X40 Z-30 F100; (Absolute command)

or

G01 U20 W-30 F100; (Incremental command)

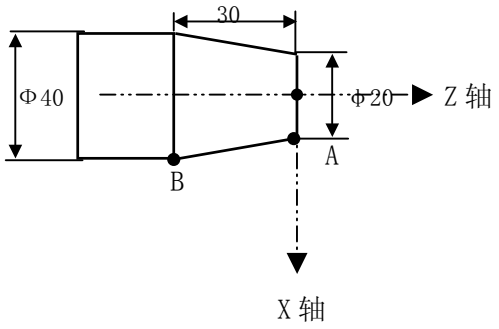


Figure 3－2 Linear Interpolation

The G01 command can specify the Linear Interpolation along one of the two axises.

The feedrate override can be adjusted by pressing the feedrate override key on the panel. The adjustable scope is (0% ~ 150%).

G01 is equivalent to G1.

(3) Circular Interpolation(G02, G03)

The commands below will move a tool along a circular arc.

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_ ;

Description of the Command Format

ITEM	Data to be given		COMMAND	MIEANING
1	Rotation direction		G02	Clockwise direction (CW)
			G03	Counterclockwise direction (CCW)
2	End point position	Absolute Value	X、Z	End point position in the work coordinate system
		Incremental Value	U、W	Distance from start point to end point
3	Distance from start point to center		I、K	I indicates X_p axis distance from the start point to the center of an arc. K indicates Z_p axis distance.
	Radius of arc		R	Radius of arc.
4	Feedrate		F	Feedrate along the arc

The CW and CCW is defined in right hand coordinate system.See Figure 3-3.

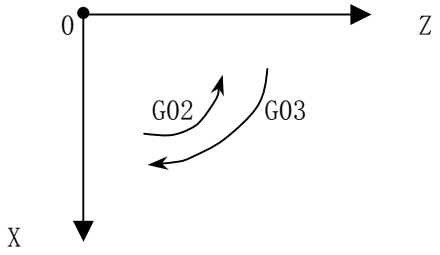


Figure 3－3 Circular Interpolation

The end point position is specified by address X, Z or U, W. The address X,Z specify End point position in the work coordinate system. The address U,W specify distance from start point to end point. The arc center is specified by addresses I and K for the X and Z axes, respectively. The numerical value following I or K, however, is a vector component in which the arc center is seen from the start point, and is always specified as an incremental value as shown in Figure 3－4. I and K must be signed according to the direction

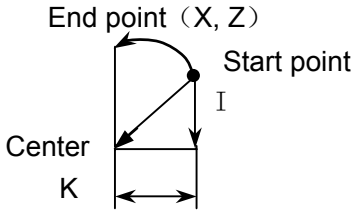


图 3－4 The center position of arc

Except by I and K, the arc center can also be specified by radius R. If so ,the arc cannot be more than 180 degree. An arc more than 180 degree can only be specified by I and K.

Example:

Diameter programmimg . The tool moves from A to B along an arc.The radius of the arc, R=25 .See Figure 3-5

G02 X50 Z-20 I10 K-5；or

G02 U20 W-20 I10 K-5；or

G02 X50 Z-20 R25；or

G02 U20 W-20 R25；

The feedrate for circular interpolation is specified by F code and the feedrate along the arc (the tangential feedrate of the arc) is controlled to be the specified feedrate.

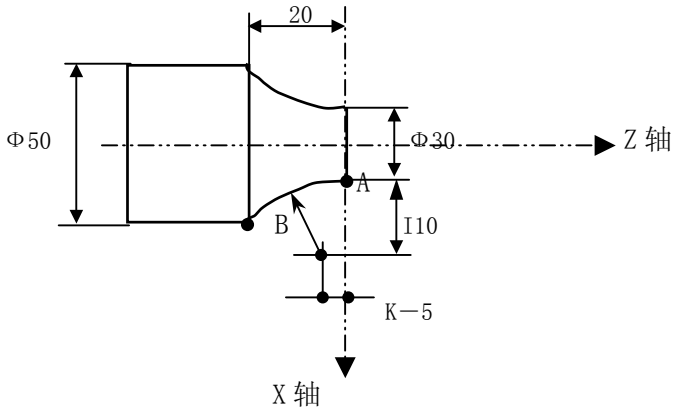


Figure 3－5 Circular Interpolation

Note 1: When either I or K is zero, the word can be omitted.

Note 2: The error between the specified feedrate and the actual tool feedrate is 2% or less. However, this feed rate is measured along the arc after the tool nose radius compensation is applied.

Ntoe 3: If I, K, and R addresses are specified simultaneously, the arc specified by address R takes precedence and the other are ignored.

Note 4: If the radius at the start point is not equal to the one at the end point and the center is specified in the address R, an alarm will be output .However if the center is specified in the address I and K, no alarm wii be output. So the address I and K is used more frequently.

(4) Thread Cutting (G32)

Tapered screws and scroll threads in addition to equal lead straight threads can be cut by using a G32 command.

Format:

G32 X（U）__ Z（W）__ F / I__；

The end point is specified by coordinate position(X,Z) in the work coordinate system or by move distance (U,W) from the start point to end point. In Straight threads cutting,X（U）should be omitted . In scroll threads cutting ,Z（W）should be omitted .In tapered screws cutting, neither X（U）and Z（W）should be omitted.

By feeding the tool synchronizing with the spindle rotation, thread cutting of the specified lead is performed. Spedify the lead of the long axis direction with the F code. F specify the metric system, and I specify the inch system.

F: The lead in metric system. Unit: mm

I: The lead in inch system. Unit: Tooth number per inch.

In general, thread cutting is repeated along the same tool path in rough cutting through finish cutting for a screw. Since thread cutting starts when the position coder mounted on the spindle outputs a 1–turn signal, threading is started at a fixed point and the tool path on the workpiece is unchanged for repeated thread cutting. Note that the spindle speed must remain constant from rough cutting through finish cutting. If not, incorrect thread lead will occur.

At the beginning and the end part of the thread cutting, the lead would have an error because of acceleration. Considering the effect, the command thread length should be longer than that needed.

Example:

1: Straight thread cutting (See Figure 3－6）

Thread lead：4mm

＆1＝3mm

＆2＝1.5mm

Depth of cut :1mm (cut twice)

(Metric input, Diameter programming)

Program as shown below：

……

```
G00 U-62.0；
G32 W-74.5 F4.0；
G00 U62；
W74.5；
U-64；(For the second cut, cut 1mm more)
G32 W-74.5；
```

G00 U64.0；
W74.5；
……

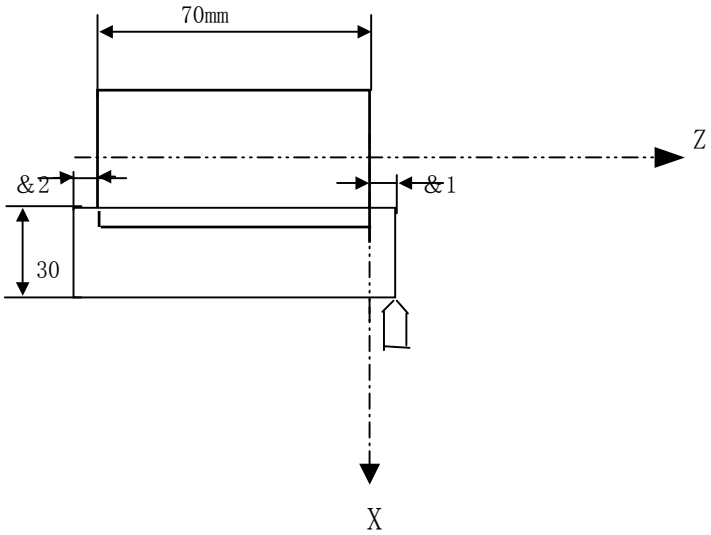
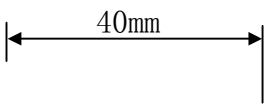


Figure 3－6 Straight thread cutting

2 : Tapered thread cutting (See Figure 3－7）

Thread lead : 3.5mm in the direction of the Z axis
&1＝3mm ,&2＝1.5mm
Cutting depth in the X axis direction is 1mm
(Cut twice)
(Metric input, Diameter programming)。
Program as shown below：

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



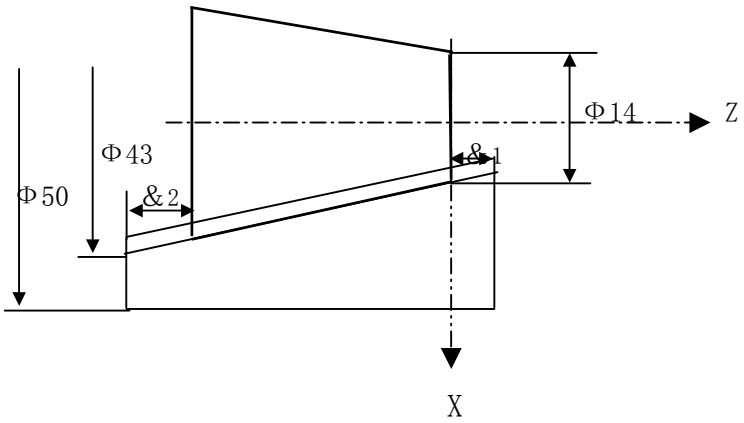


Figure 3－7 Tapered thread cutting

Note 1: Feedrate override is ineffective (fixed at 100%) during thread cutting.

Note 2: It is very dangerous to stop feeding the thread cutter. This will suddenly increase the cutting depth. Thus, the feed hold function is ineffective while thread cutting. If the feed hold button is pressed during thread cutting, the tool will stop after a block not specifying thread cutting is executed as if the SINGLE BLOCK button were pushed.

Note 3: When thread cutting is executed in the single block status, the tool stops after execution of the first block not specifying thread cutting.

Note 4: When the previous block was a thread cutting block, cutting will start immediately without waiting for detection of the 1–turn signal even if the present block is a thread cutting block.

Example:

G32 W-20 F3 (A 1–turn signal is detected before this block.)

G32 W-30 F3 (A 1–turn signal is not detected before this block.)

(5) Dwell---G04

By specifying a dwell, the execution of the next block is delayed by the specified time.

Format:

G04 P_; or G04 X_; or G04 U_ ;

X_ , U_ and P_ can Specify a time. The command value range by X and U is 0.001~ 99999.999 second and decimal point is permitted. The command value range by P is 1~ 99999999 millisecond and decimal point is not permitted.

Example:

G04 X 1; Dwell 1 second

G04 P1000; Dwell 1 second

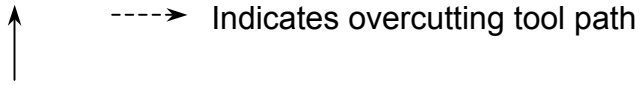
G04 U 1; Dwell 1 second

Special Usage:

X,U and P are all omitted.Here G04 stops the tool in position specified by previous block.By this way, overcutting can be avoided in sharp-angled tool path.

（20，10）

Insert G04 here



Example：……

N150 G01 X20 Z10 F100；

N160 G04；（Avoid overcutting）

N170 G01 W-10；

……

(6) Automatic Reference Position Return(G28)

The reference position is a fixed position on a machine tool to which the tool can easily be moved by the reference position return function. Tools are automatically moved to the reference position via an intermediate position along a specified axis.

Format:

G28 X/U_ Z/W_ ;

An intermediate position is specified by (X/U , Z/W) .

The process of Reference position return is shown in Figure 3-8 .

(1) Rapid traverse and position at the intermediate position of the command axes(A position→B position).

(2) Rapid return from the intermediate position to the reference position (B position→R position).

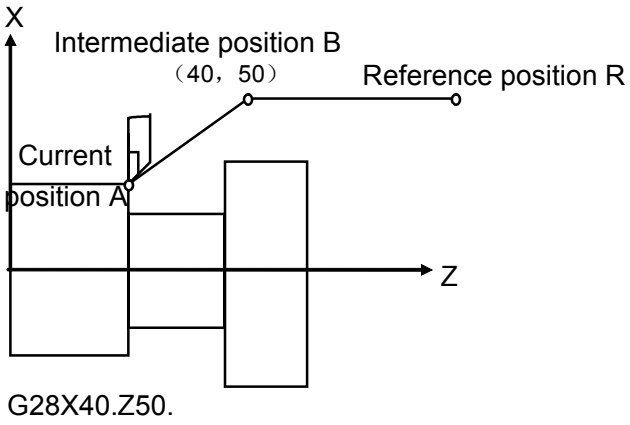


Figure 3-8 The motion of the reference position return

Note 1: When the G28 command is specified when manual return to the reference position has not been performed after the power has been turned on, the movement from the intermediate point is the same as in manual return to the reference position.

(7) Coordinate system setting(G50)

Format:

G50 X(x) Z(z) ;

A workpiece coordinate system is set so that a point on the tool, such as the tool tip, is at specified coordinates(X , Z). If a coordinate system is set using G50 during offset, a coordinate system in which the position before offset matches the position specified in G50 is set.

Refer Chapter 4 for detail.

3.3 Feed function

Feedrate of linear interpolation (G01), circular interpolation (G02, G03), etc. are commanded with numbers after the F code.

Format: **F××；**

Cutting feed is controlled so that the tangential feedrate is always set at a specified feedrate. A common upper limit can be set on the cutting feedrate along each axis with parameter (P25). If an actual cutting feedrate (with an override applied) exceeds a specified upper limit, it is clamped to the upper limit. An upper limit is set in mm/min . An override from 0% to 150% (in 10% steps) can be applied to feedrate with the key on the machine operator's panel.

Two modes of specification are available:

1. Feed per minute (G98)

After F, specify the amount of feed of the tool per minute. The unit is mm/minute.

After specifying G98, the amount of feed of the tool per minute is to be directly specified by setting a number after F. G98 is a modal code. Once a G98 is specified, it is valid until G99 (feed per revolution) is specified. At power–on, the feed per revolution mode is set.

2. Feed per revolution (G99)

After F, specify the amount of feed of the tool per spindle revolution.The unit is mm/revl.

After specifying G99, the amount of feed of the tool per spindle revolution is to be directly specified by setting a number after F. G99 is a modal code. Once a G99 is specified, it is valid until G98 (feed per minute) is specified

3.4 Miscellaneous function (M functions)

When a move command and miscellaneous function are specified in the same block, the commands are executed simultaneously.

When address M followed by 2 digit value is specified, a code signal and strobe signal are transmitted. These signals are used for turning on/off the power to the machine. In one program block, only one M code can be specified.

3.4.1 General M code：

M03： spindle positive revolution

M04： spindle reversal

M05： spindle stop

M08： cooling fluid on

M09： cooling fluid off

M32： lubricating on

M33： lubricating off

M98/M99 :Calling of sub program/End of sub program

M00： Program pauses, and can be restarted by actuating the cycle operation.

M30： Automatic operation is stopped and contol returns to the start of the program.

Except M00 and M30, the executing time of the other M code can be specified with the parameter №035.The unit is ms.

The following M codes indicate special meaning:

(1)M30 (End of program)

1) This indicates the end of the main program.

2) Cycle operation is stopped and the NC uint is reset.

3) Contol returns to the start of the main program.

4) The spindle and cooling fluid can be stopped simultaneously.

(2) M02:(End of program)

1) Except remaining the output of M codes unchanged, others are the same as M30.

2) This command must be specified in a sigle block.

(3) M00:(Program stop)

Automatic operation is stopped after a block containing M00 is executed. When the program is stopped, all existing modal information remains unchanged. The automatic operation can be restarted by actuating the cycle operation.

(4) M98/M99(Calling of subprogram / End of subprogram)

See the subprogram section for details .

Note 1: Even if there is a program block following M00 or M30, it cannot be executed.

Note 2: When executing M98 and M99, code singal is not transmitted.

3.5 Spindle Functions (S code)

3.5.1 Spindle speed command

The spindle speed can be controlled by specifying a value following address S. A block can contain only one S code. When a move command and an S code command are in the same block , the commands are executed simultaneously. The spindle speed command can be specified in two specification. In the first specification the spindle speed is specified with a binary code, and in the second specification the spindle speed value is specified directly.

(1) S 2-digit command

This spindle speed can be specified by address S followed by 2–digit code. In the 2–digit specification, the parameter SANG/P004 should be cleared to ‘0’.

The binary codes including S01, S02, S03, S04 are output directly from the port XS57 of KND1TBⅡ CNC, which can control four stage variable speed.

Related parameters:

0	4	5	STIME1
0	4	6	STIME2

STIEM1：When a new binary code is specified ,the delay time 1：0~4080毫秒。

STIEM2：When a new binary code is specified ,the delay time 2：0~4080毫秒。

For example:The procedure of control the spindle speed from S01 to S02.The time set by STIEM1 is delayed after S01 is switched off .Then, S02 is switched on and after the time set by STIEM2 the next program block is executed.

(2) S 4-digit command（collocating transducer）

The spindle speed can be specified directly by address S followed by a 4–digit value (rpm). In the 4–digit specification, the parameter SANG/P004 should be set to ‘1’. The upper limit spinle speed may vary depending on the machine tool builder. The value may be set by parameter P31. If the spindle speed specified in program is over the upper limit in P31,the actual speed is clamped at the upper limit.

In the 4–digit specification, the spindle speed is controlled by the voltage output from the analog spindle port.

Note ： The ahead two kinds of collocation are specified in the bit parameter SANG in the parameter P004.

3.5.2 Constant Surface Speed Control (G96, G97)

Specify the surface speed (relative speed between the tool and workpiece) following S. The spindle is rotated so that the surface speed is constant regardless of the position of the tool.Voltage is fed to the spindle control section so that the spindle rotate to produce the correct surface speed.

The unit of surface speed is m/min .

(1) Command Format

Constant surface speed controlling command:

G96 S__ ; Specify surface speed after S (m/min)

Constant surface speed controlling cancel command:

G97 S__ ; Specify spindle speed after S (rpm)

(2) Clamp of maximum spindle speed

Format:

G50 S_ ; The maximum spindle speed (rpm) follows S.

When constant surface speed control is applied, a spindle speed higher than the value specified in G50S_; (maximum spindle speed) is clamped at the maximum spindle speed.

Note1：In a rapid traverse block specified by G00, the constant surface speed control is not made by calculating the surface speed to a transient change of the tool position, but is made by calculating the surface speed based on the position at the end point of the rapid traverse block. In the cutting block specified by G01, G02 and G03 etc. , the constant surface speed control is effective.

Note 2：The value for S specified in G96 mode is also used in G97 mode and is restored when returning to G96 mode.

G96 S50；（50 m/min）

G97 S100；（1000 rpm）

G96 G01 X100；（50 m/min）

Note 3：When switching from G96 to 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.

N100 G97 S800；（800 rpm）

N200 G96 S100；（100 m/min）

N300 G97；（X rpm）

X is the spindle speed in the block before N300.When the G96 mode changes to G97 mode ,the spindle speed does not change.When G97→G96, the value of S in the G96 mode is effective. If S is not specified yet, S = 0 m/min .

Note 4：Even when a machine is operating in machine lock status, the constant surface speed is calculated according to the change in the coordinate value of the x axis in the program.

Note 5：The constant surface speed control is also effective during threading. Accordingly, it is recommended that the constant surface speed control be invalidated with G97 command before starting the scroll threading and taper threading, because the response problem in the servo system may not be considered when the spindle speed changes.

Note 6：G99 and G96 can be effective at the same time.

Note 7：The CNC calculates the spindle speed which is proportional to the specified surface speed at the position of the programmed coordinate value on the X axis. This is not the value calculated according to the X axis coordinate after offset when offset is valid.

Example：See Figure 3－9

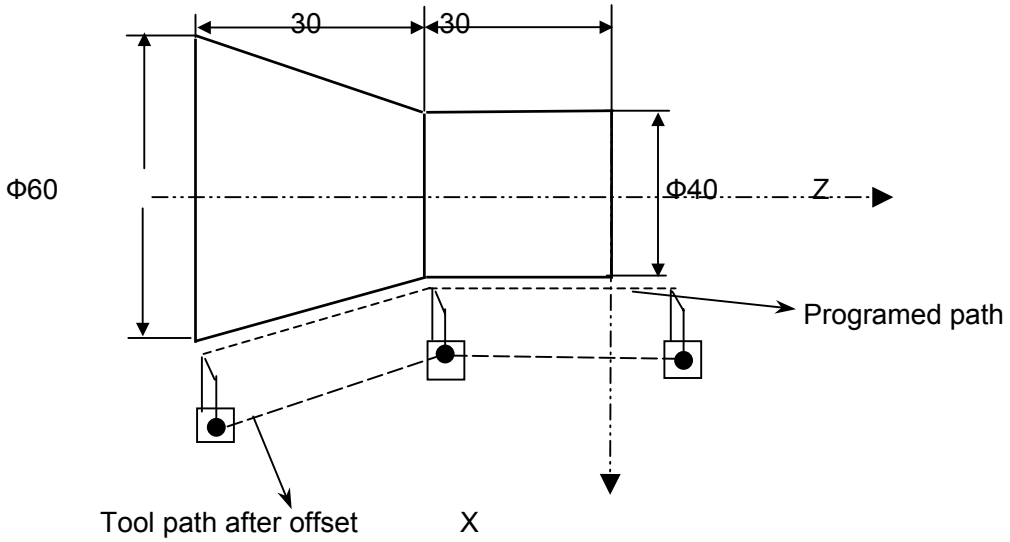


Figure 3－9 Constant surface speed control

Program：

N10 ……;

NG00 X100 Z80；

N40 T0101；

N50 X40 Z10；

N60 G50 S2000； （ Designation of max. spindle speed）

N70 G96 S200； （ Surface speed 200 m/min ）

N80 G01 Z-30 F100；

N90 X60 Z-60；

N100 Z－65；

N110 G97 S500；

N120 ……;

3.5.3 Chuck Control function

（1）Parameter enable the function

The chuck control function is effective if QPSL/P036 is set to 1

0	3	6								
						QPSL				

（2）Relate parameters

0	4	1								
			QPLS	QPM3						

QPLS 0: The signal fed to the chuck control section is Voltage level.

1: The signal fed to the chuck control section is Voltage pulses.The period of pulse is set with P051.

QPM3 0: If the chuck's grip is lose before the spindle's startup, an alarm is output and the operation is stopped.

1: No check is done before the spindle's startup.

0	4	3								
			QPIN							

QPIN 0: No signal is fed about the chuck's grip is lose or not.

1: The signal is fed about the chuck's grip is lose or not.Check it before the spindle's startup.

0	5	1	QPLSTIME

QPLSTIME：The period of the chuck control pulse. The unit is ms.

（3）Chuck control command(M10/M11)

M10 ：Chuck clamp

M11 ：Chuck lose

（4）DI/DO signal

DI signal

0	0	0								
						QPI				

QPI 1：Manual lose chuck signal

0	0	3								
			QPJI	QPSI						

QPJI 1：Indicates the chuck is clamping

QPSI 1：Indicates the chuck is lose

DO signal

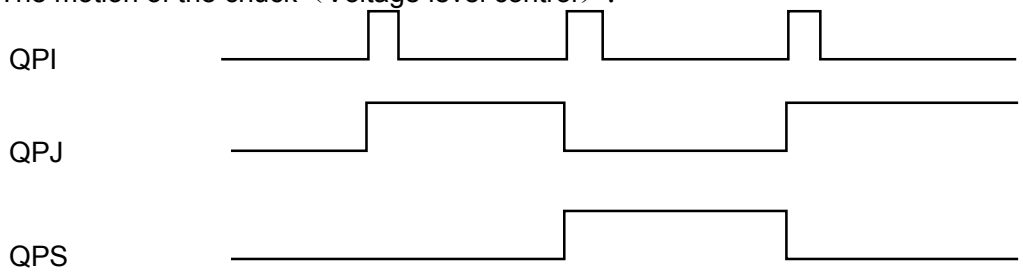
0	0	5

QPJ	QPS						

QPJ: This signal is fed to clamp the chuck

QPS: This signal is fed to lose the chuck

The motion of the chuck（Voltage level control）：



The signal QPJ and QPS are all zero at power-on.

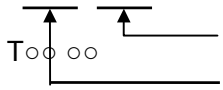
The QPI signal is ineffective during the spindle is rotating.

3.6 Tool Functions

3.6.1 TOOL SELECTION

By specifying a 2–digit/4–digit numerical value following address T, a code signal and a strobe signal are transmitted to the machine tool. This is mainly used to setect tools on the machine. When a move command and a T code are specified in the same block, the commands are executed simultaneously.

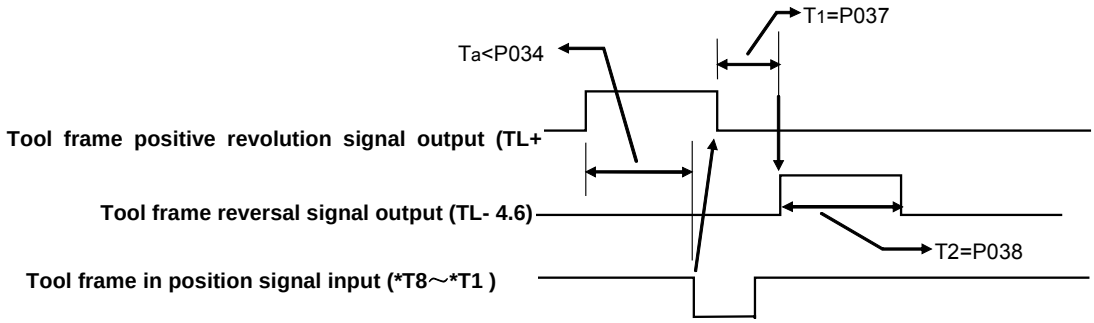
First two digits of T code indicates the desired tool. Last two digits of T code is used as the offset number indicating the compensation amount for tool offset.



Tool offset number

Tool selection

(1) Tool selection process



When Ta≥P034, The alarm 05 is output to indicate that the time of tool selection is too long.

When beginning to execute T code, output tool carrier positive revolution signal (TL+) at first, and make tool carrier rotate; when the tool carrier position signal of the T code is received, the position revolution signal turns off. After delaying for the period of time specified by parameter P37, the tool carrier begins to reversely rotate to be locked (TL-), whose width is decified by parameter P038, then turn off the reversal signal, and tool change is finished. The next program block is executed in seccession. If the specified tool number is the same as the current one, the tool change command ends at once, and the next program is executed.

(2) Related parameters

Tool position signal（*T8～*T1） is specified to positive pulse or negative pulse with Bit1 TSGN of the parameter P003.

TSGN 0： Tool position singal is effective when the pulse is positive.（constantly turn on）

1： Tool position singal is effective when the pulse is negative.（constantly turn off）

T1: The delaying time from tool carrier positive revolution stop to its reversal locked singal output beginning.

P037：0～4080 ms（desingation unit：ms，interval unit：16ms）

Ttool number ：tool number selecting

P039：designation value 0～8 unit：entries

T2 ：the pulse width of tool carrier reversal locked singal

P038： 0～4080 ms（designation：ms，interval unit：16ms）

Ta ：the longest time of tool change and position: 0～1000S。

P034 （designation unit：ms，interval：16ms）

Note：The ahead parameter cannot be specified with 0.

(3) Alarm：

1） 03：T code error

When the tool number specified by T code is more than the maximal number specified by №039, the alarm will give birth to, and stop tool change and machining program at the same time.

2） 05：tool change time too long

If the specified tool position is still not get after the delaying time,specified by P034, from the tool position revolution beginning, the alarm will give birth to, and stop tool change and machining program at the same time.

About the relation between the command tool number in program block and the actual tool, please refer to the specification of the manufactory.

(4) Timing scaning and checking the tool carrier signal

When the bit7 (CKTDI) of the parameter P036 is 1, the system scans and checks the tool carrier signal at certain interval, and finishes the following function:

- After finishing tool change, check the tool carrier signal once more. If it is rignt, end tool change; or alarm, and pause program executing. (give birth to pause signal)
- Check the tool carrier signal at certain interval, and judge it whether the same as the system recording.
- Checking content :1 what should be put through is so or not 2 what should not be put through is neither or not. The error of these two circs will give birth to alarm.

08：the total numbers of tool position error or tool input signal error

Note 1：Check the number of corresponding input signal according to the number of tools specified by the parameter P039.

Note 2：If it isn't need to check or group tool is used, specify CKTDI=0.

(5) Tail tool carrier selection

When using back tool carrier, specify the bit parameter of P036: RVX=1.

Note ： when RVX=1, the manual direction of X axes specified anteriorly is reverted.

3.6.2 Compensation Functions

Tool offset is used to compensate for the difference when the tool actually used differs from the imagined tool used in programming (usually, standard tool).

In this system, tool offset is controlled only by T code, and not controlled by G code.

T code's meaning:

T ○○ ○○

Tool offset number

Tool number

A) Tool selection

Tool selection is realized by specifying the T code corresponding to tool number. About the relation between tool number and tool, please refer to the specification of the manufactory.

B) Tool offset number

It is use to select the offset value. The offset value must be specified by MDI/LCD cell. Every offset number have two offset value, one for X axes, the other for Z axes.

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

The range of offset value is －0999.999～999.999mm

Note 1：Only T code

When only a T code is specified in a block , the tool is not moved by the offset until a move commnad is specified in the following block.The movement is performed at rapid traverse rate in G00 mode and at feedrate in other mode .

Note 2：G50 X（x） Z（z） T ； Tool is not moved.The coordinate system in which the coordinate of the tool position is (x,z) is set.The tool position is obtained by subtracting the offset value corresponding to the offset number specified in the T code.

Note 3：When only a T code is specified in a block , the tool is not moved by the offset until a move commnad is specified in the following block.However the tool can move by the offset at once if the words “U 0 W 0” are specified in the block.

3.6.3 Offset

The tool offset can be set in two ways, absolute input and increment input.Refer Chapter 4 for details.

3.7 Functions to Simplify Programming

For repetitive machining peculiar to turning, such as metal removal in rough cutting, a series of paths usually specified in a range of three to several dozen blocks, can be specified in one block. In addition , only the values to be exchanged need to be specified for repetition, and programs using this cycle are very simple and useful.

3.7.1 Canned cycles (G90, G92, G94)

There are three canned cycles : the outer diameter/internal diameter cutting canned cycle (G90), the thread cutting canned cycle (G92), and the end face turning canned cycle (G94).

(1) Outer diameter/internal diameter cutting canned cycle (G90)

1) Straight cutting cycle (for diameter programming)

Format: G90X(U)__ Z(W)__ F__;

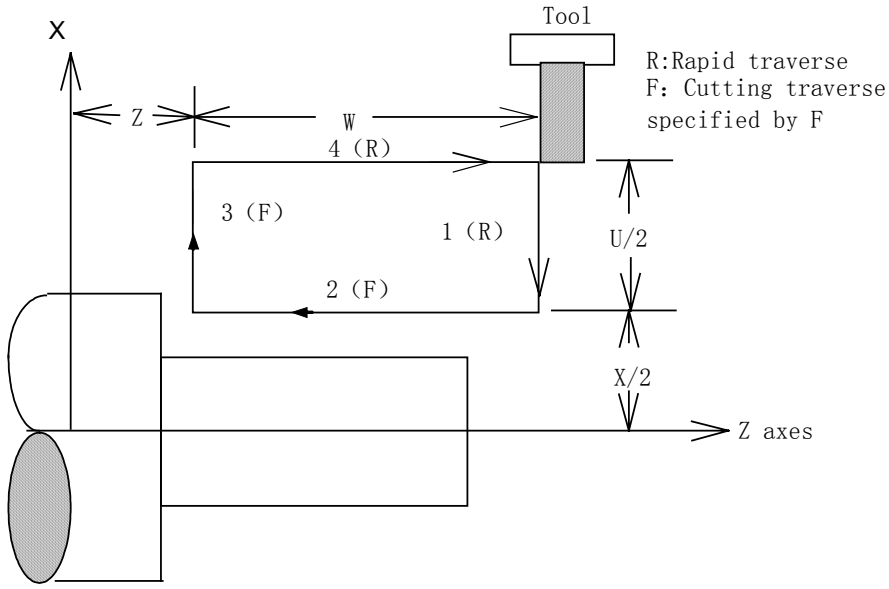


Fig.3-7-1 Straight Cutting Cycle

In incremental programming, the sign of the numbers following address U and W depends on the direction of paths 1 and 2. In the cycle of 13.1.1 (a), the signs of U and W are negative.

In single block mode, operations 1, 2, 3 and 4 are performed by pressing the cycle start button once.

2) Taper cutting cycle (for diameter programming)

Format: G90X(U)__ Z(W)__ R__ F__ ;

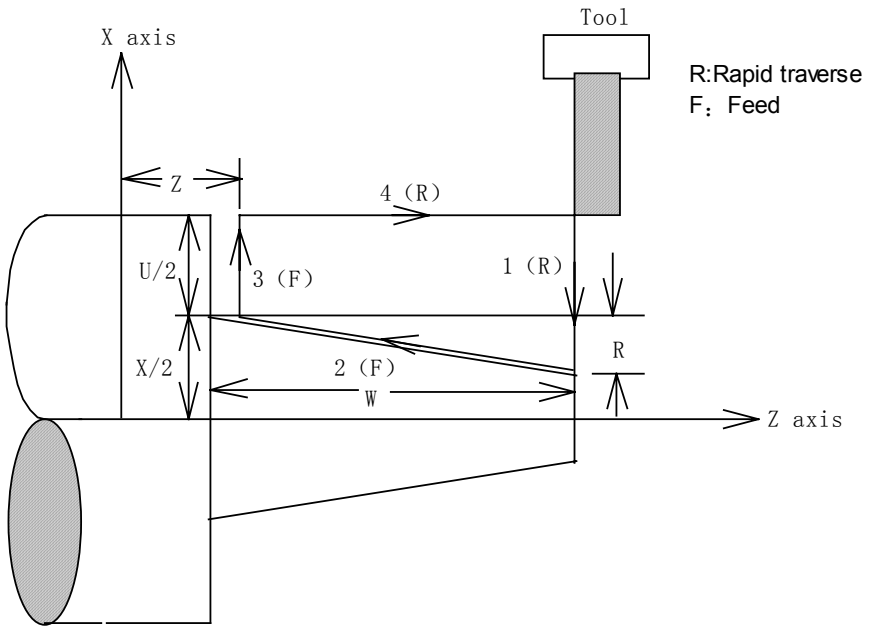
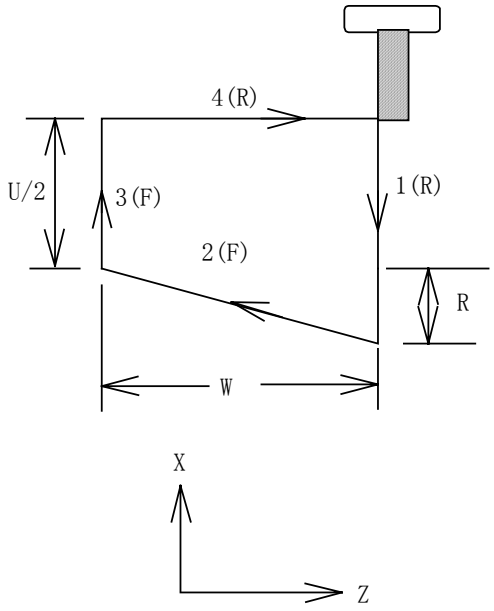


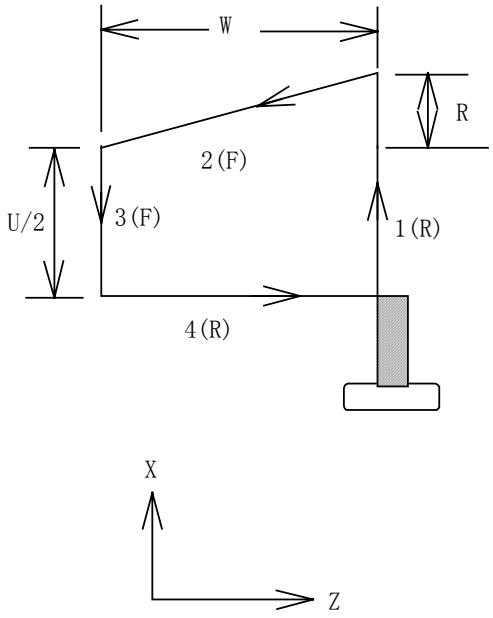
Fig. 3-7-2 Taper cutting cycle

In incremental programming, the relationship between the signs of the numbers following address U, W, and R, and the tool paths are as follows:

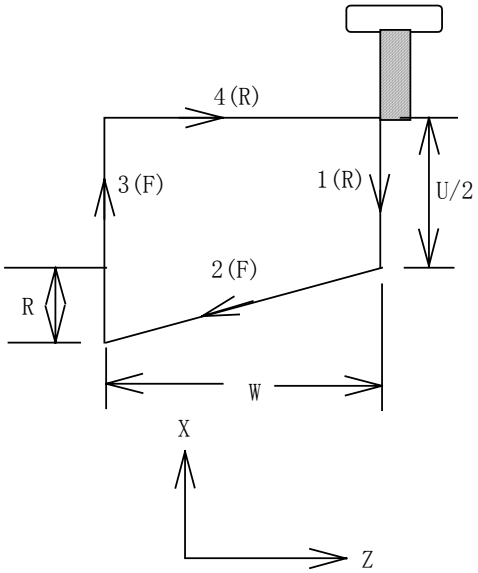
1) U<0, W<0, R<0



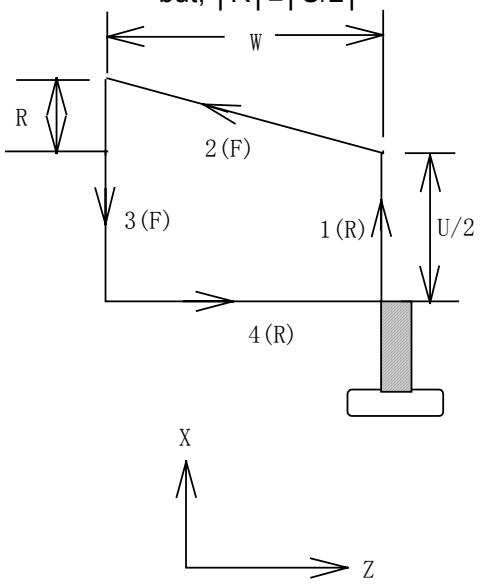
2) U>0, W<0, R>0



3) U<0, W<0, R>0
but, |R|≤|U/2|



4) U>0, W<0, R<0
but, |R|≤|U/2|



(2) Thread cutting cycleG92

1) Straight thread cutting cycle (for diameter programming)

Format: G92X(U)__ Z(W)__ F/I__ ;

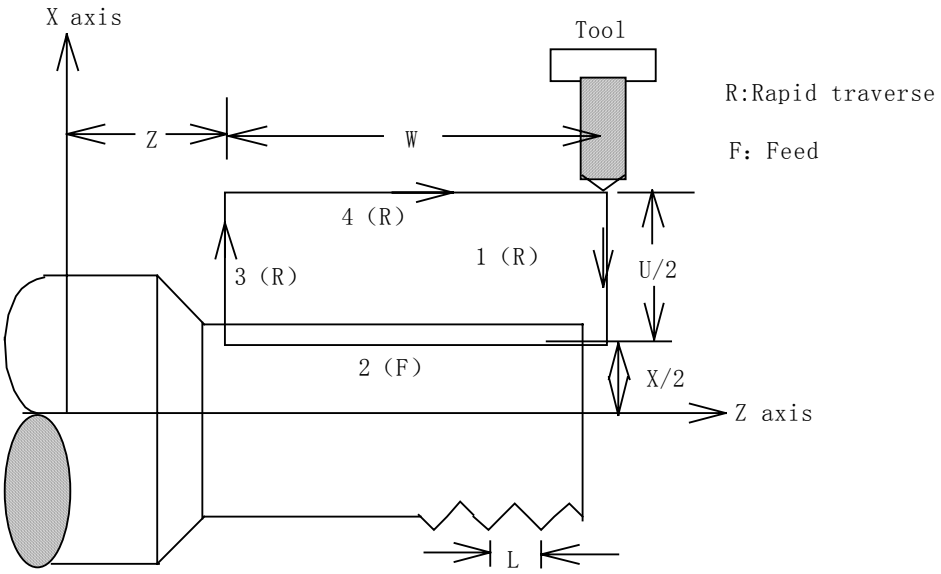


Fig. 3-7-3 Straight Thread Cutting

In incremental programming, the sign of numbers following addresses U and W depends on the direction of paths 1 and 2. That is , if the direction of path 1 is the negative along the X axis, the value of U is negative. The range of thread leads, limitation of spindle speed, etc. are the same as in G32 (thread cutting). In the single block mode, operations 1, 2, 3, and 4 are performed by pressing cycle start button once.

Note : This thread cutting are the same as in thread cutting in G32. However, a stop by feed hold is as follows ; Stop after completion of path 3 of thread cutting cycle.

2) Taper thread cutting cycle (for diameter programming)

Format: G92 X(U)__ Z(W)__ R__ F/I__ ;

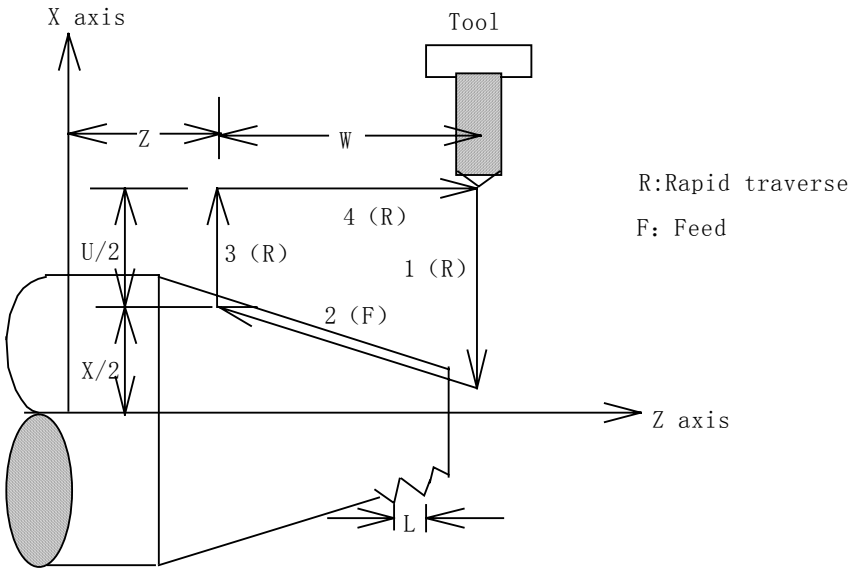


Fig. 3-7-4 Taper thread cutting cycle

3) Chamfering in thread cutting canned cycle

Format: G92 X（U）_Z（W）_ F/I _ P_ ;

P：Chamfering distance

Designation unit : 0.1 pitch

Designation range :1～255 The designation value exceed it is invalidation.

Note 1：The chamfering distance specified by P is modal. If a new value is specified ,the parameter P'28 is updated automatically so that the latest chamfering distance is effective at poweron .

Note 2：The chamfering amount is held when the system turning off.

(3) Endface Cutting cycle (G94)

1) Face cutting cycle (for diameter programming)

Format: G94 X(U)__ Z(W)__ F__ ;

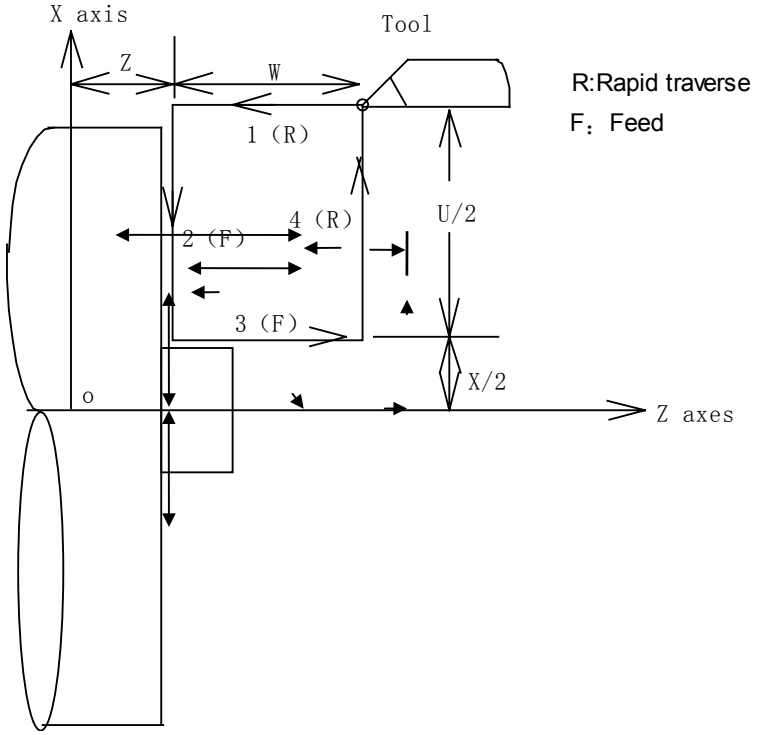


Fig. 3-7-5 Endface Cutting cycle

In incremental programming, the sign of numbers following addresses U and W depends on the direction of paths 1 and 2. That is, if the direction of the path is in the negative direction of the Z axis, the value of W is negative.

In single block mode, operations 1, 2, 3, and 4 are performed by pressing the cycle start button once.

2) Taper face cutting cycle (for diameter programming)

Format: G94 X(U)__ Z(W)__ R__ F__ ;

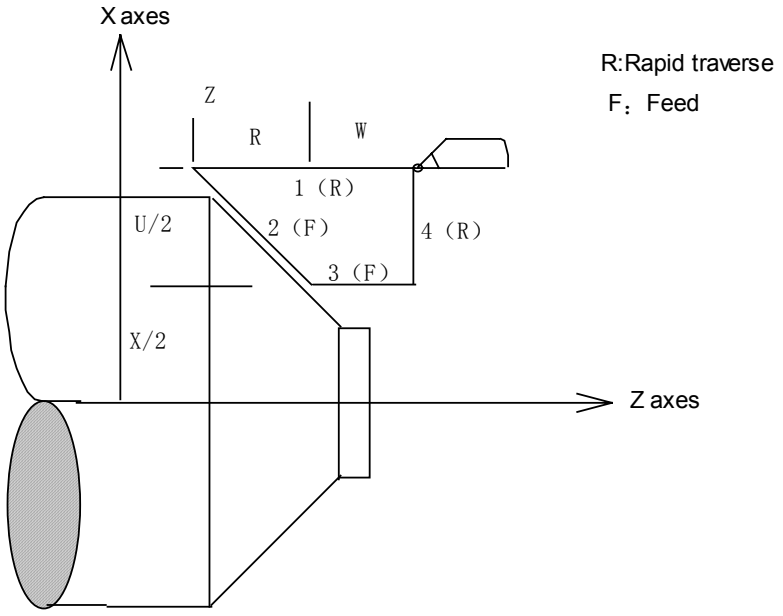
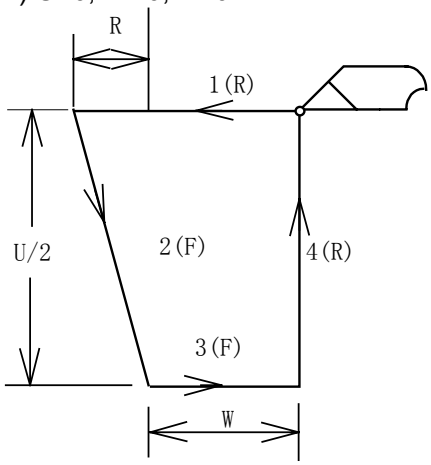


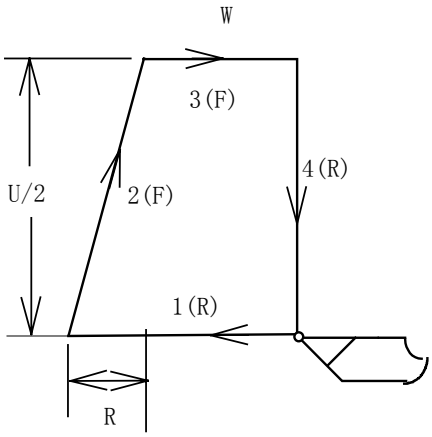
Fig. 3-7-6 Taper face cutting cycle

In incremental programming, the relationship between the signs of the numbers following address U, W, and R, and the tool paths are as follows:

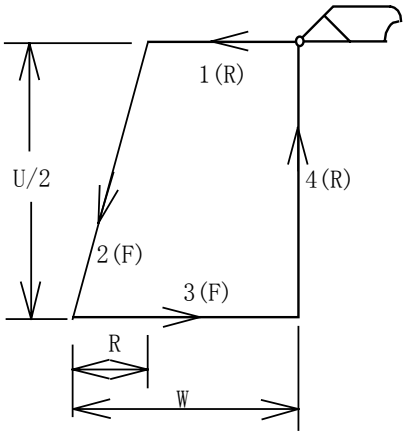
1) U<0, W<0, R<0



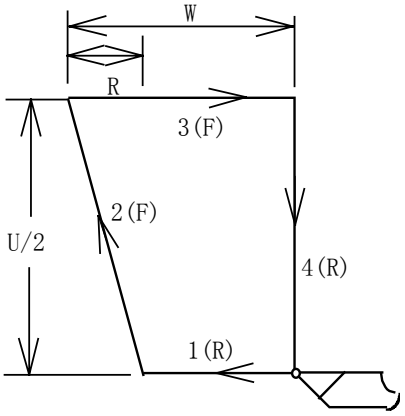
2) U>0, W<0, R<0



(3) U<0, W<0, R>0 (｜R｜≤｜W｜)

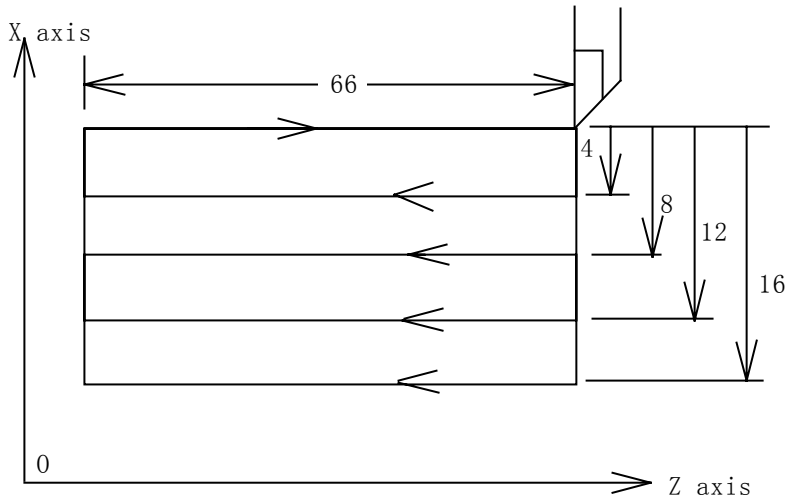


4) U>0, W<0, R>0(｜R｜≤｜W｜)



Note1: Since data values of X (U), Z (W) and R during canned cycle are modal, if X (U), Z (W), or R is not newly commanded, the previously specified data is effective. Thus, when the Z axis movement amount does not vary as in the example below, a canned cycle can be repeated only by specifying the movement commands for the X–axis. However, these data are cleared, if a G code in the group 01 except for G90, G92,G94 is commanded.

(example)



The cycle in the above figure is executed by the following program.

```
N030 G90 U-8000 W-66000 F4000 ;
N031 U-16000 ;
N032 U-24000 ;
N033 U-32000 ;
```

Note 2: The following three applications can be performed.

(1) If an EOB or zero movement commands are specified for the block following that specified with a canned cycle, the same canned cycle is repeated.

(2) By specifying a canned cycle in the MDI mode, and pushing the cycle start button after the block terminates, the same canned cycle as the previous one will be performed.

(3) If the M, S, T function is commanded during the canned cycle mode, both the canned cycle and M, S, or T function can be performed simultaneously. If this is inconvenient, cancel the canned cycle once as in the program examples below (specify G00 or G01) and execute the M, S, or T command. After the execution of M, S, or T terminates, command the canned cycle again.

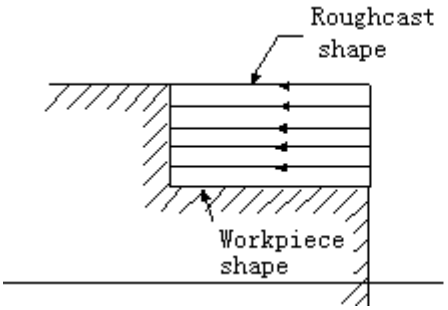
(example)

```
N003 T0101 ;
....
....
N010 G90 X20000 Z10000 F2000 ;
N011 G00 T0202 ;
N012 G90 X20500 Z10000 ;
```

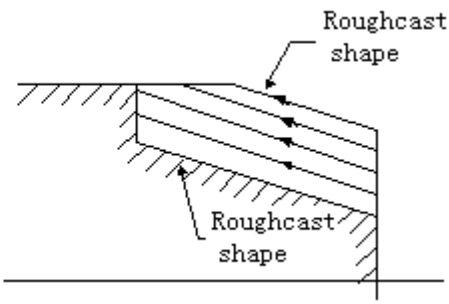
（4）How to Use Canned Cycles (G90, G92, G94)

An appropriate canned cycle is selected according to the shape of the material and the shape of the product.

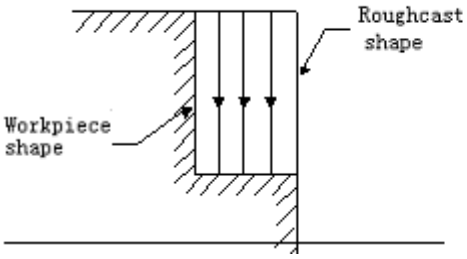
(1) Straight cutting cycle



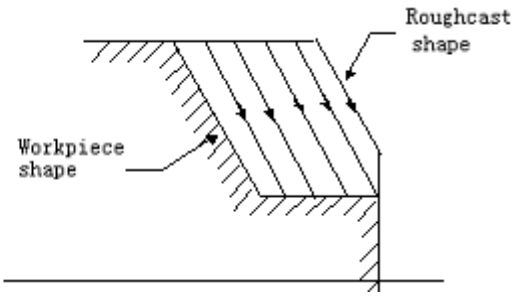
(2) Taper cutting cycle



(3) Cutting cycle in face



(4) Tapping cycle in face



3.7.2 Multiple cutting canned cycle (G93)

3.7.3 MULTIPLE REPETITIVE CYCLE (G70~G76)

This option canned cycles to make CNC programming easy. For instance, the data of the finish work shape describes the tool path for rough machining. And also, a canned cycles for the thread cutting is available.

(1) Stock Removal in Turning (G71)

If a finished shape of A to A' to B is given by a program as in the figure below, the specified area is removed by △d (depth of cut), with finishing allowance △u/2 and △w left.

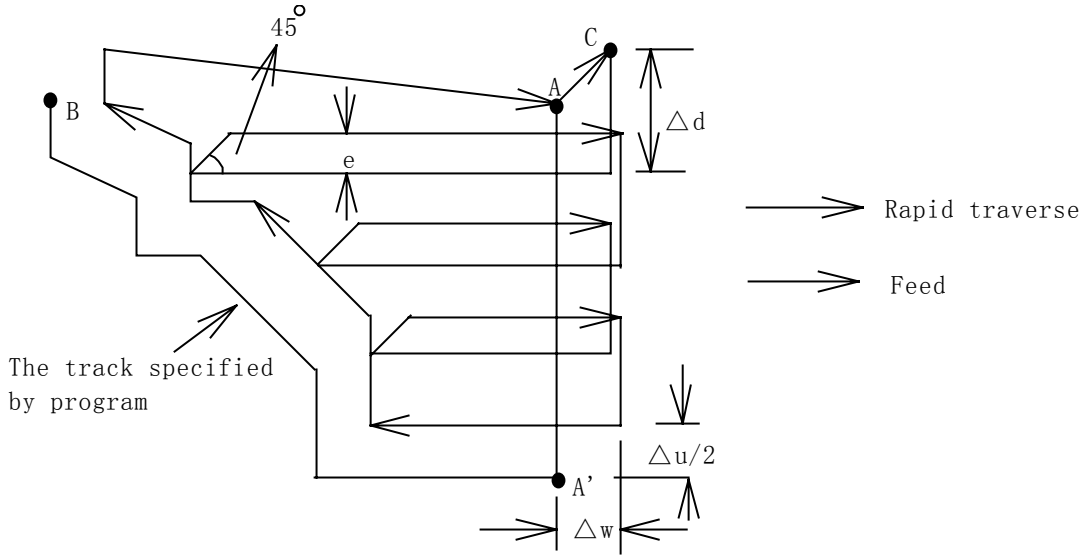


Fig. 3-7-7 Stock Removal in Turning

Format:

G71 U（△d） R(e);

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

N(**ns**) . . . . . .

. . . . . .

· F

· S

· T

.

.

N(nf). . . . . .

The move command between A and B is specified in the blocks from sequence number ns to nf.

d _ Depth of cut (radius designation)

Designate without sign. The cutting direction depends on the direction AA'. This designation is modal and is not changed until the other value is designated. Also this value can be specified by the parameter P21,and the parameter is changed by the program command.

e : Escaping amount

This designation is modal and is not changed until the other value is designated. Also this value can be specified by the parameter P22, and the parameter is changed by the program command.

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

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

u : Distance and direction of finishing allowance in X direction (diameter designation).

w : Distance and direction of finishing allowance in Z direction.

F,S,T: In the G71 cycle, functions F,S,T are invalidation in the program blocks between NS～NF, and are all omitted. The are valid only in the program blok G71.

Note 1: The cycle machining is performed by G71 command with P and Q specification. F, S, and T functions which are specified in the move command between points A and B are ineffective and those specified in G71 block or the previous block are effective.

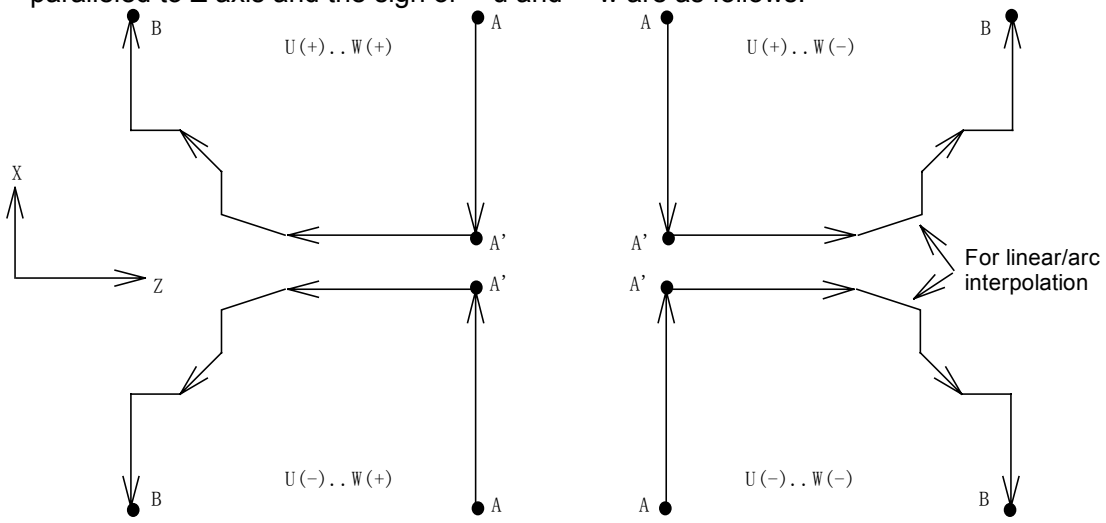
Note 2: The tool path between A' and B must show monotone increase or decrease along the X axis.

Note 3: When an option of constant surface speed control is selected, G96 or G97 command specified in the move command between points A and B are ineffective, and that specified in G71 block or the previous block is effective.

Note 4: 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 Z axis cannot be specified.

Note 5: The subprogram cannot be called from the block between sequence number "ns" and "nf".

Note6: The following four cutting patterns are considered. All of these cutting cycles are made paralleled to Z axis and the sign of u and w are as follows:



(2) Stock Removal in Facing (G72)

As shown in the figure below, this cycle is the same as G71 except that cutting is made by a

operation parallel to X axis.

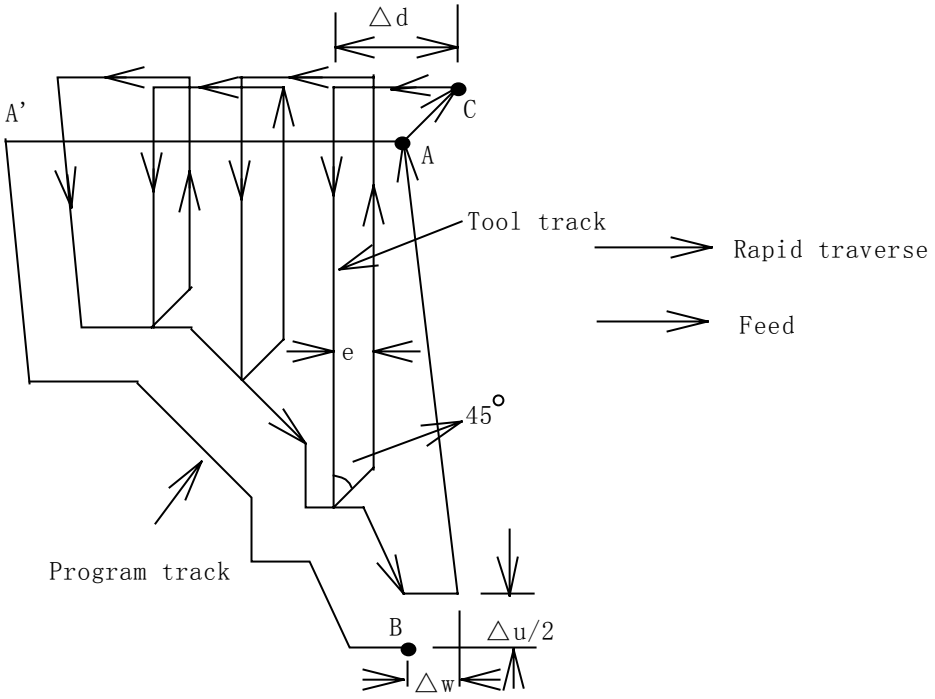


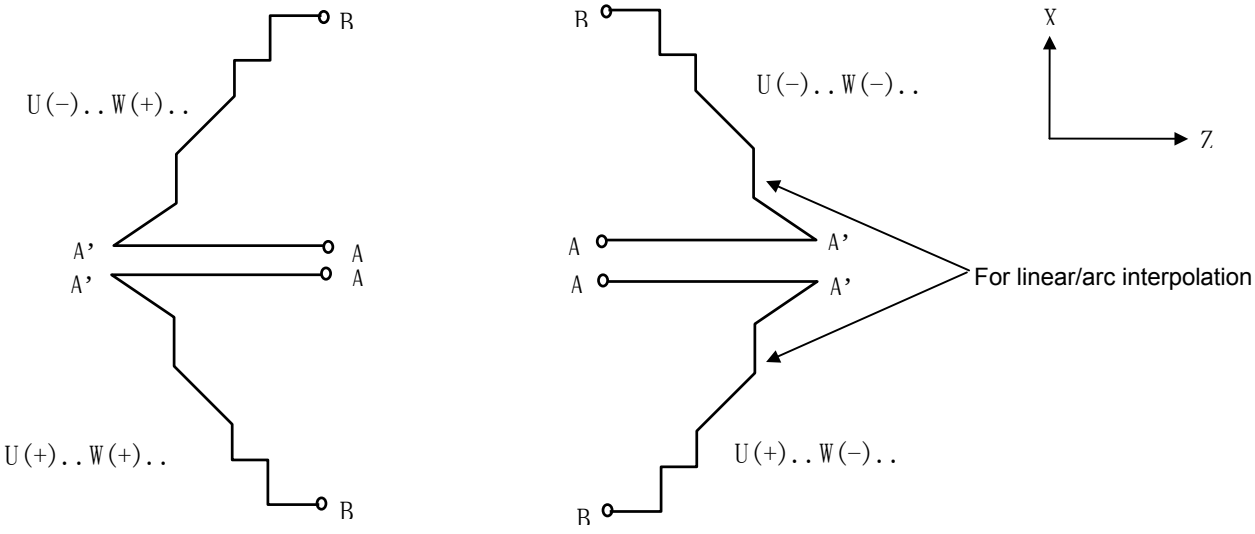
Fig. 3-7-8 Stock Removal in Facing

G72 W(d) R(e) ;

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

The meanings of d, e, ns, nf, u, w, f, s, and t are the same as those in G71.

The following four cutting patterns are considered. All of these cutting cycles are made parallel to X axis and the sign of u and w are as follows:



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 axes.

(3) Pattern Repeating (G73)

This function permits cutting a fixed pattern repeatedly, with a pattern being displaced bit by bit. By

this cutting cycle, it is possible to efficiently cut work whose rough shape has already been made by a rough machining, forging or casting method, etc.

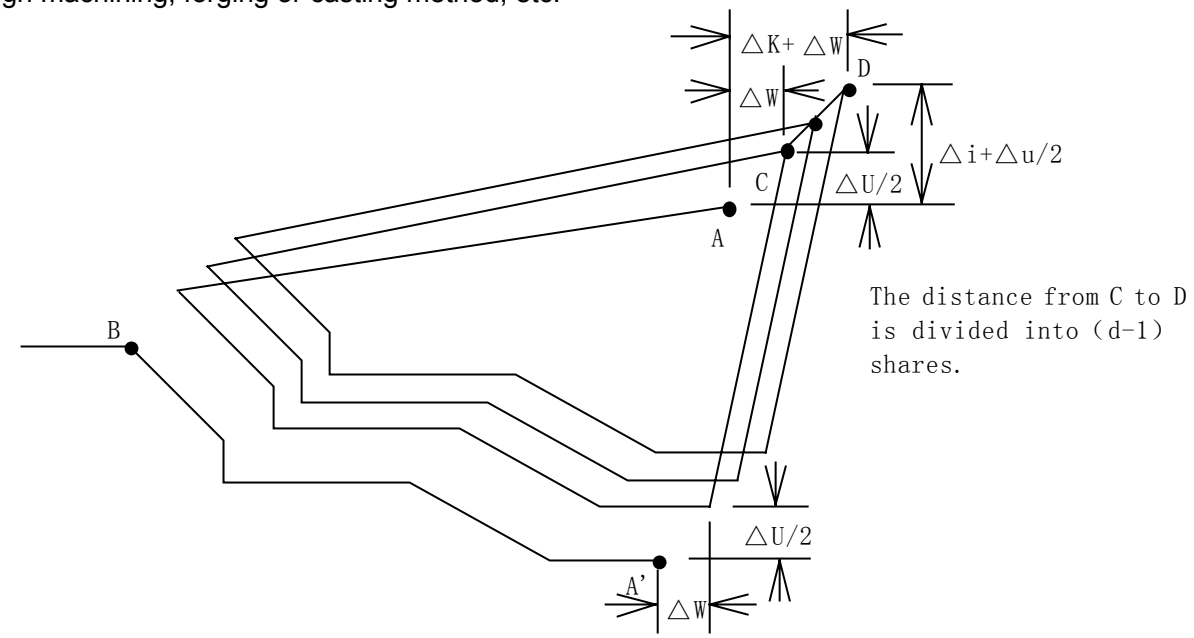


Fig. 3-7-9 Pattern Repeating

The pattern commanded in the program should be as follows.

A →A'→B

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

G73 P (ns) Q (nf) U (u) W (w) F (f) S (s) T (t) _

N(ns) . . . . . .

. . . . . .

.

.

.

N(nf) . . . . . .

The move command between A and B is specified in the blocks from sequence number ns to nf.

i : Distance and direction of relief in the X axis direction (Radius designation). This designation is modal and is not changed until the other value is designated. Also this value can be specified by the parameter P23, and the parameter is changed by the program command.

k : Distance and direction of relief in the Z axis direction. This designation is modal and is not changed until the other value is designated. Also this value can be specified by the parameter P24, and the parameter is changed by the program command.

d : The number of division This value is the same as the repetitive count for rough cutting. This designation is modal and is not changed until the other value is designated. Also, this value can be specified by the parameter P25, and the parameter is changed by the program command.

Example:

The word R0.001 indacates that the number of division is 1;

The word R1 indacates that the number of division is 1000;

The meanings of ns, nf, u, w, f, s, and t are the same as those in G71. Any F, S, and T function contained in the blocks between sequence number ”ns” and ”nf” are ignored, and the F, S, and T functions in this G73 block are effective.

Note 1: The cycle machining is performed by G73 command with P and Q specification. The four cutting patterns are considered. Take care of the sign of u, w, k, and i. When the

machining cycle is terminated, the tool returns to point A.

(4) Finishing Cycle (G70)

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

G70 P(ns) Q(nf) ;

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

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

F, S, and T functions specified in the block G71, G72, G73 are not effective but those specified between sequence numbers ”ns” and ”nf” are effective in G70. When the cycle machining by G70 is terminated, the tool is returned to the start point and the next block is read.

(example: 1) Usage of G70 and G71

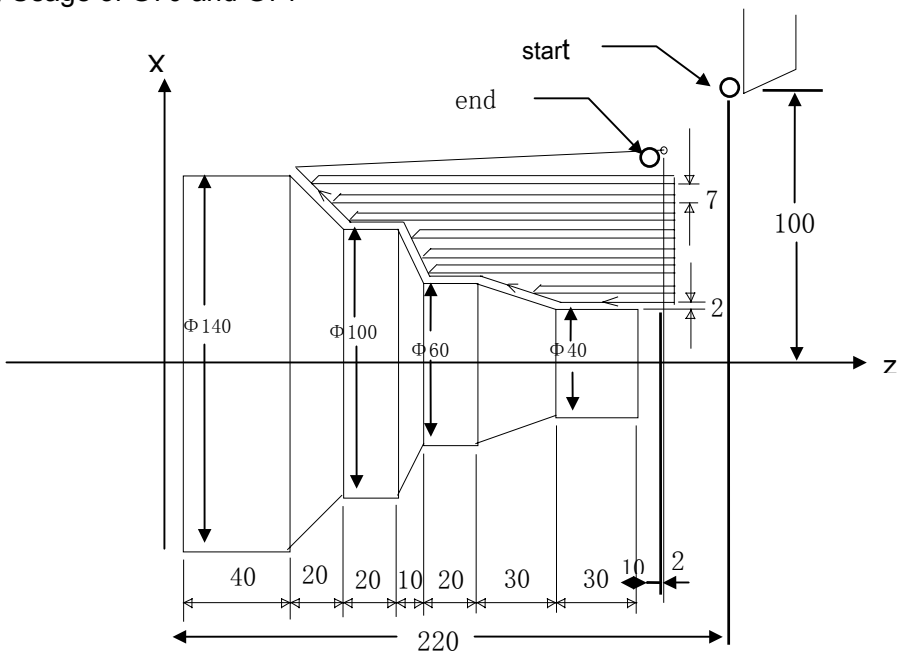


Fig. 3-7-10 Usage of G70 and G71

(diameter designation, metric input)

```
N010 G50 X200.0 Z220.0 ;
N011 G00 X160.0 Z180.0 ;
N012 G71 U7.0 R1.0 ;
N013 G71 P014 Q020 U4.0 W2.0 F0.3 S55 ;
N014 G00 X40.0 F0.15 S58 ;
N015 G01 W-40.0 ;
N016 X60.0 W-30.0 ;
N017 W-20.0 ;
N018 X100.0 W-10.0 ;
N019 W-20.0 ;
N020 X140.0 W-20.0 ;
N021 G70 P014 Q020 ;
```

(example: 2) Usage of G70 and G72

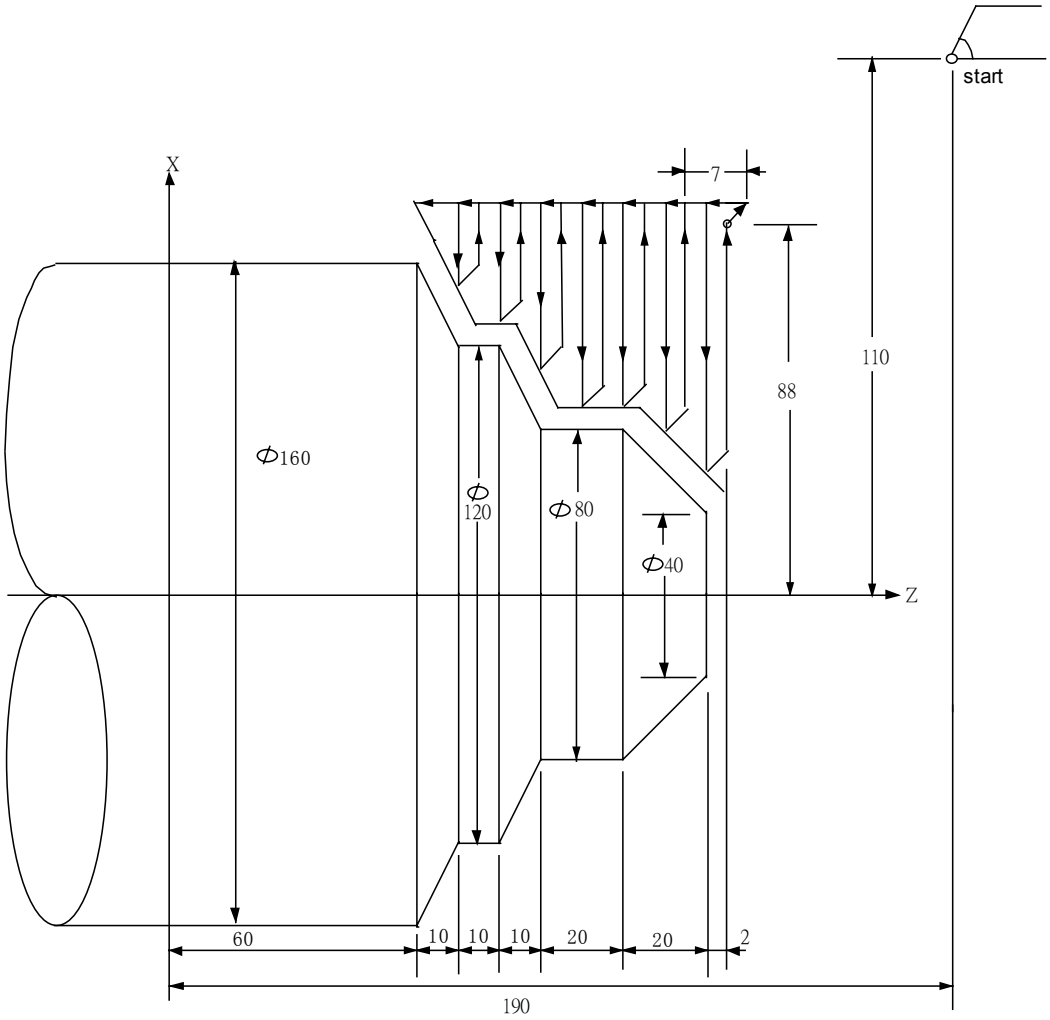


Fig. 3-7-11 Usage of G70 and G72

(diameter designation, metric input)

```
N010 G50 X220.0 Z190.0 ;
N011 G00 X176.0 Z132.0 ;
N012 G72 W7.0 R1.0 ;
N013 G72 P014 Q019 U4.0 W2.0 F0.3 S55 ;
N014 G00 Z58.0 S58 ;
N015 G01 X120.0 W12.0 F0.15 ;
N016 W10.0 ;
N017 X80.0 W10.0 ;
N018 W20.0 ;
N019 X36.0 W22.08
N020 G70 P014 Q019 ;
```

(example: 3) Usage of G70 and G73

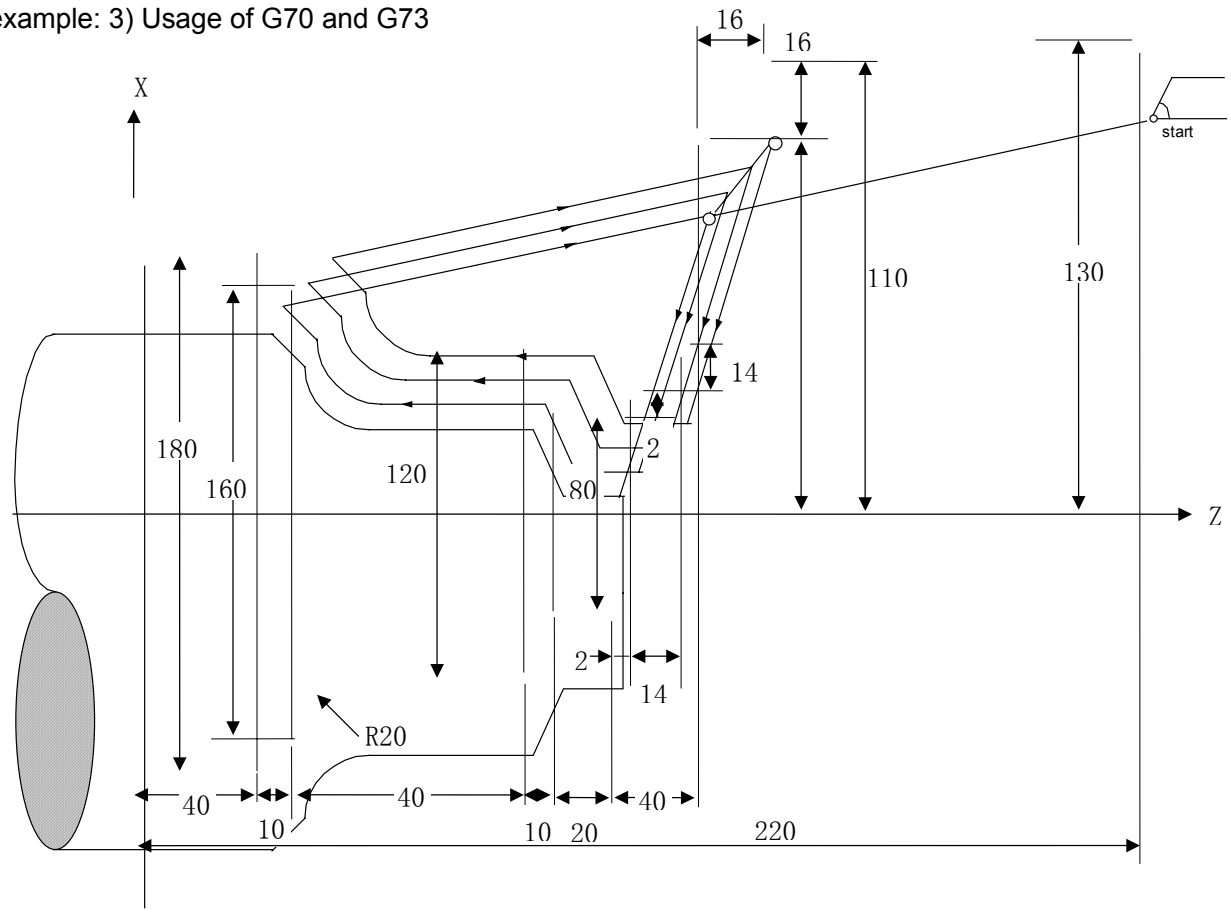


Fig. 3-7-12 Usage of G70 and G73

(diameter designation, metric input)

```
N010 G50 X260.0 Z220.0 ;
N011 G00 X220.0 Z160.0 ;
N012 G73 U14.0 W14.0 R3 ;
N013 G73 P014 Q019 U4.0 W2.0 F0.3 S0180 ;
N014 G00 X80.0 W-40.0 ;
N015 G01 W-20.0 F0.15 S0600 ;
N016 X120.0 W-10.0 ;
N017 W-20.0 S0400 ;
N018 G02 X160.0 W-20.0 R20.0 ;
N019 G01 X180.0 W-10.0 S0280 ;
N020 G70 P014 Q019 ;
```

(5) End Face Peck Drilling Cycle (G74)

This command is used for drilling.

Format：

G74 R（e）；

G74 Z（w） Q（△k） F（f）；

e: Return amount

This designation is modal and is not changed until the other value is designated. Also this value can be specified by the parameter P26 ,and the parameter is changed by the program

command.

Z： Z component of point C

W： Increment amount from A to C

Δk： Depth of cut in Z direction (without sign)

F： Cutting feedrate.

The tool path is showed in Fig. 3-7-13 When G74 is executed.

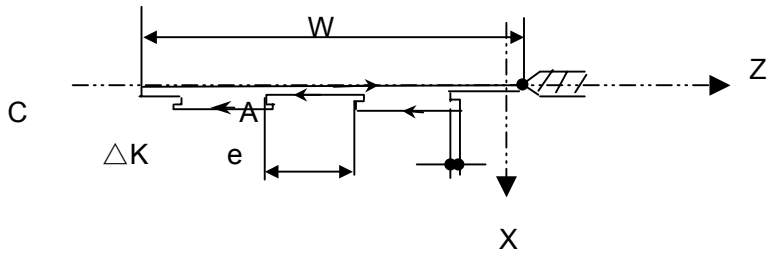


Fig. 3-7-13 Cutting Path in End Face Peek Drilling Cycle

Example：See Fig. 3-7-14

N10 G00 X0 Z10；

N20 G74 R2；

N30 G74 Z-80 Q10000 F800；

N40 G00 X50 Z50；

N50 M30；

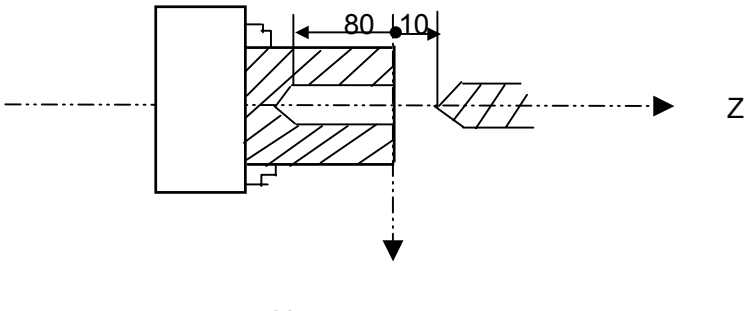


Fig. 3-7-14 Usage of G74

(6) Outer Ddiameter Drilling Cycle (G75)

The following command can move a tool as shown in Fig. 3-7-15. This is equivalent to G74 except that X is replaced by Z. Chip breaking is possible in this cycle, and grooving in X axis and peck drilling in X axis (in this case, Z, W, and Q are omitted) are possible.

Format:

G75 X(U)__ P(Δi)__ F(f)__;

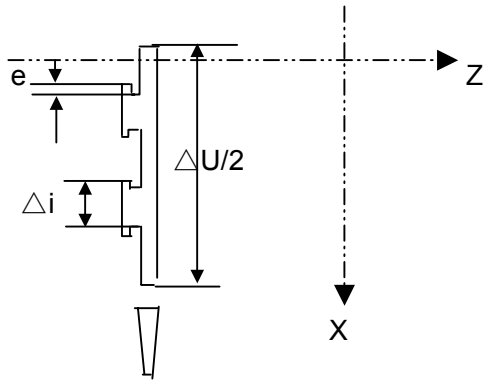
e ： : Return amount

X（U）： Depth of the groove

△i ： Depth of cut in X direction (without sign)

f ： Feedrate

The tool path is shown below:



Example：See Fig.3-7-15

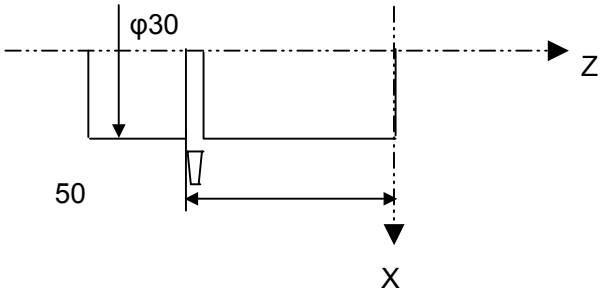


Fig. 3-7-15 Usage of G75

Program：

O0004；

N10 M03 S××;

N20 T0101;

N30 G00 X35 Z-50;

N40 G75 R1;

N50 G75 X-1 P5000 F60;

N60 G00 X100 Z50 M09;

N70 M05;

N80 T0100;

N90 M30;

(7) Multiple Thread Cutting Cycle(G76)

The thread cutting cycle as shown in Fig. 3-7-16 is programmed by the G76 command.

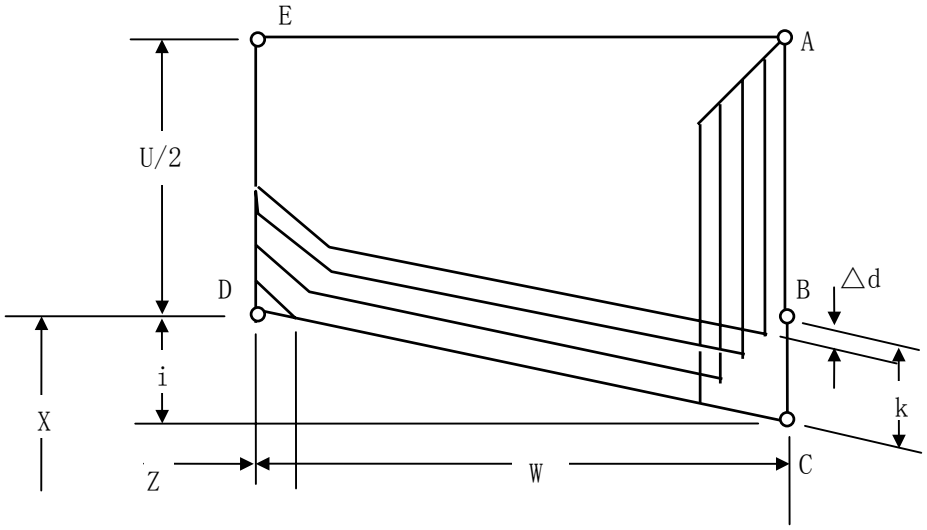
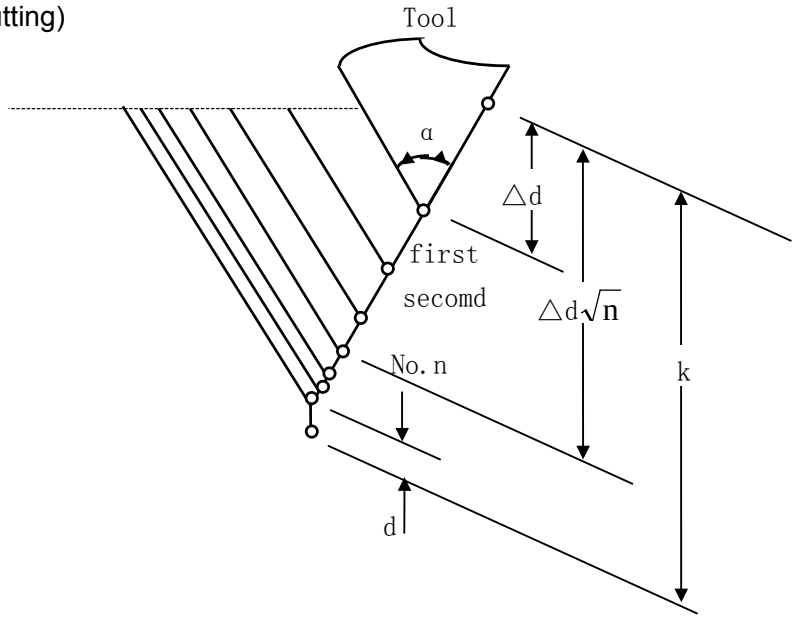


Fig. 3-7-16 Multiple Thread Cutting Cycle

(the detail of cutting)



G76 P(m) (r) (a) Q(Δdmin) R(d) ;

G76 X(U) Z(W) R(i) P(k) Q(Δd) F(L) ;

m: Repetitive count in finishing (1 to 99)

This designation is modal and is not changed until the other value is designated. Also this value can be specified by the parameter P27, and the parameter is changed by the program command.

r : Chamfering amount

When the thread lead is expressed by L, the value of r can be set from 0.0L to 9.9L in 0.1L increment (2–digit number from 00 to 90). This designation is modal and is not changed until the other value is designated. Also this value can be specified by the parameter P28, and the parameter is changed by the program command.

a : Angle of tool tip

One of six kinds of angle, 80 , 60 , 55 , 30 , 29 , and 0 _ can be selected, and specified by 2–digit number. This designation is modal and is not changed until the other value is designated. Also this value can be specified by the parameter P29, and the parameter is changed by the program command.

m, r, and a are specified by address P at the same time.
(Example)
When m=2, r=1.2L, a=60°, specify as shown below (L is lead of thread).
P 02 12 60
m r a
dmin : Minimum cutting depth (specified by the radius value)

When the cutting depth of one cycle operation ($\Delta D \times\sqrt{N}$ -$\Delta D\times\sqrt{N-1}$) becomes smaller than this limit, the cutting depth is clamped at this value. This designation is modal and is not changed until the other value is designated. Also this value can be specified by parameter P30, and the parameter is changed by the program command.

d : Finishing allowance (Command with radius amount)

This designation is modal and is not changed until the other value is designated. Also this value can be specified by parameter P31, and the parameter is changed by the program command.

i : Taper value command with radius amount

If i = 0, ordinary straight thread cutting can be made.

k : Height of thread

This value is specified by the radius value.

d : Depth of cut in 1st cut (Command with radins amount)

L : Lead of thread (same as G32).

Note 1: The cycle machining is performed by G76 command with X (U) and Z (W) specification. By using this cycle , one edge cutting is performed and the load on the tool tip is reduced.

Making the cutting depth d for the first path, and $d\times\sqrt{N}$ for the nth path, cutting amount per one cycle is held constant.

Four symmetrical patterns are considered corresponding to the sign of each address. The internal thread cutting is available. In the above figure, the feed rate between C and D is specified by address F, and in the other path, at rapid traverse.

The sign of incremental dimensions for the above figure is as follows :

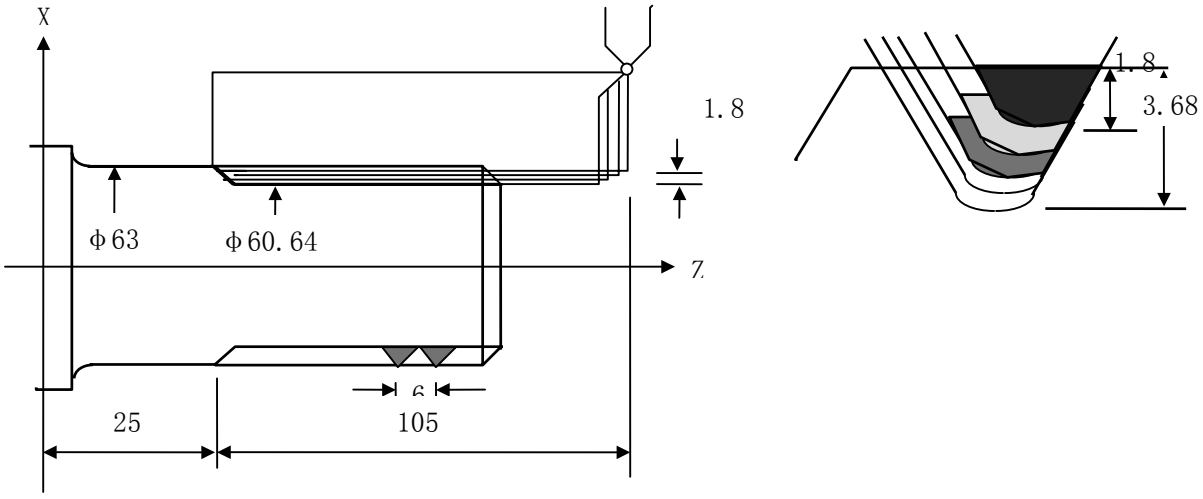
U, W : minus (determined by the direction of the tool path AC and CD.)

i : minus (determined by the direction of the tool path AC.)

k : plus (always)

Δd : plus (always)

(example) multiple canned cycle (G76)



G00 X80 Z130 ;
G76 P011060 Q100 R200 ;
G76 X60640 Z25000 P3680 Q1800 F6.0 ;

(8) Notes on Multiple Repetitive Cycle (G70 to G76)

(1) In the blocks where the multiple repetitive cycle are commanded, the addresses P, Q, X, Z, U, W, and R should be specified correctly for each block.

(2) In the block which is specified by address P of G71, G72 or G73, G00 or G01 group should be commanded. If it is not commanded, P/S alarm No.65 is generated.

(3) In MDI mode, G70, G71, G72, or G73 cannot be commanded. If it is commanded, P/S alarm No. 67 is generated. G74, G75, and G76 can be commanded in MDI mode.

(4) In the blocks in which G70, G71, G72, or G73 are commanded and between the sequence number specified by P and Q, M98 (subprogram call) and M99 (subprogram end) cannot be commanded.

(5) In the blocks between the sequence number specified by P and Q, the following commands cannot be specified.

- ✧ One shot G code except for G04 (dwell)
- ✧ 01 group G code except for G00, G01, G02, and G03
- ✧ 06 group G code
- ✧ M98 / M99

(6) While a multiple repetitive cycle (G70 to G76) is being executed, it is possible to stop the cycle and to perform manual operation. But, when the cycle operation is restarted, the tool should be returned to the position where the cycle operation is stopped. If the cycle operation is restarted without returning to the stop position, the movement in manual operation is added to the absolute value, and the tool path is shifted by the movement amount in manual operation.

(7) When G70, G71, G72, or G73 is executed, the sequence number specified by address P and Q should not be specified twice or more in the same program.

(8) Do not program so that the final movement command of the finishing shape block group designated with P and Q for G70, G71, G72, and G73 finishes with chamfering or corner rounding. If it is specified,P/S alarm No. 69 is generated.

Ⅲ OPERATION

1 General

Before operating KND1TB CNC ,the following topic should be understood well.

1.1 Manual operation

(1) Manual reference position return
(2) The tool movement by manual operation
(3) The auxiliary functions by manual operation

1.2 Automatic operation

(1) Memory operation

After the program is once registered in memory of CNC, the machine can be run according to the program instructions. This operation is called memory operation

(2) MDI operation

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

1.3 Editing a part program

(1) How to register a part program into the memory of the CNC.
(2) How to edit the program.

1.4 Testing a program

Before machining is started, the automatic running check can be executed. It checks whether the created program can operate the machine as desired. This check can be accomplished by running the machine actually or viewing the position display change (without running the machine)

A) Check by running the Machine

<1> Feedrate override

Check the program by changing the feedrate override.

<2> Single block

When the cycle start button is pressed, the machine tool will execute one block program and stop after the block is finished. The machine tool will execute next operation and stop by pressing the cycle start button again. The program is checked in this manner.

1.5 Displaying and setting data

The operator can display or change a value stored in CNC internal memory by key operation on the CRT/MDI screen

(1)How to display and set the offset value.
(2)How to display and set the parameters.
(3)How to display the status of DI/DO signals fed to the machine.

1.6 Display

(1)Program display
(2)Current position display
(3)Alarm display
(4)Graphic display

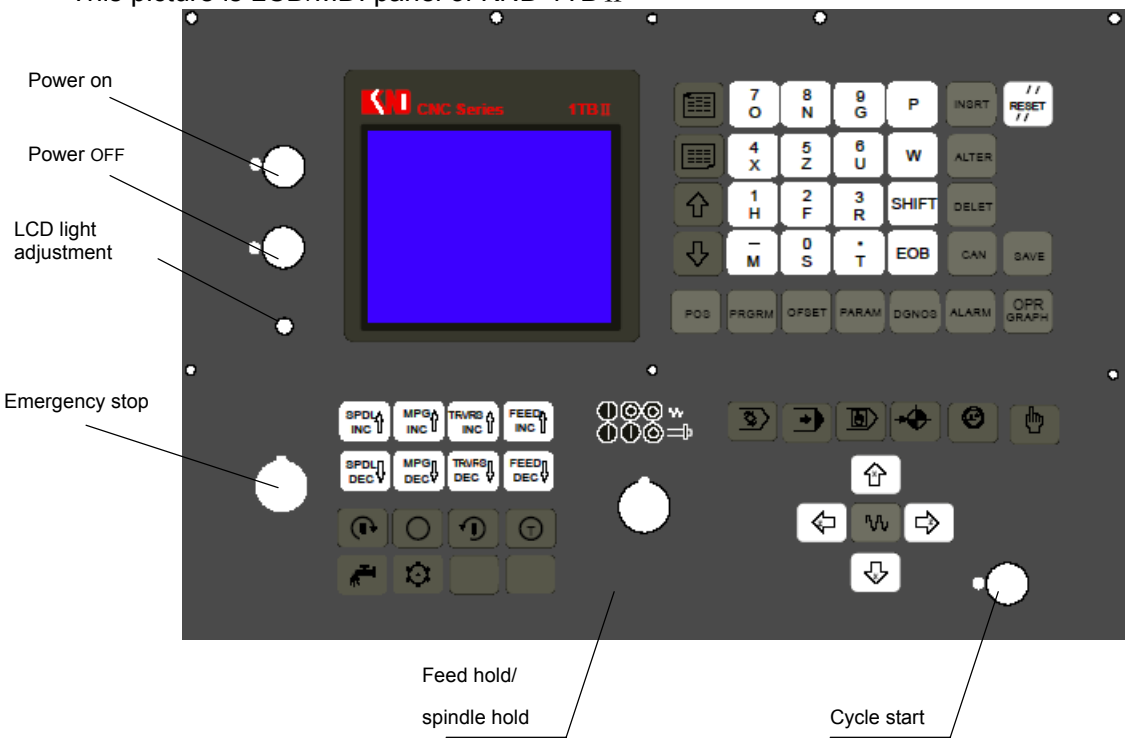
1.7 Operation of the non-voltile memory

Programs, offset values, parameters, etc. input in CNC memory can be output to the non-voltile memory for saving. After once output to the medium, the data can be input into CNC memory.

2 LCD/MDI OPERATION PANEL

2.1 LCD/MDI PANEL

This picture is LCD/MDI panel of KND 1TB II



2.2 ADJUSTMENT OF LCD LIGHT

The KND1TB CNC is designed with a 6-inch LCD. The light of LCD can be adjusted by turning the light knob of LCD.

NOTE 1:The light of LCD is sensitive to the temperature around.The best display may be acquired by turning the Light adjustment knob.

2.3 DIAPLSY FUNCTION KEY

When one of the function key is pressed, the corresponding screen is displayed.When a function key is pressed twice or third times,2nd and 3nd chapter (if they exist) of the function is respectively displayed.

[POS]: Press the key to display Position screen.
[PRGRM]: Press the key to display Program screen.
[OFSET]: Press the key to display Offset screen.
[PARAM]: Press the key to display Parameter screen.
[ALARM]: Press the key to display Alarm screen.
[OPR/GRAPH]: Press the key to display Diagnostic screen.

2.4 OPERATION MODE KEY

When one of the mode-selection key is pressed, the corresponding operation mode will be selected.All the modes are listed below.

[EDIT] [AUTO] [MDI] [ZRN] [STEP/HANDLE] [JOG]

2.5 Explanation of the MDI keyboard

Num.	Name	Explanation
1	RESET key	Press this key to reset the CNC, to cancel an alarm, etc.
2	Address and numeric keys	Press these keys to input alphabetic, numeric, and other characters.
3	Function keys	Press theses keys to switch display screens for each function.
4	Operation mode key	Press theses keys to select a operation mode
5	Cancel key	Press this key to delete the last character or symbol input to the key input buffer when the key input buffer displays.
6	Cursor move keys	There are two different cursor move keys. ↓: This key is used to move the cursor in a downward or forward direction. ↑: This key is used to move the cursor in an upward or reverse direction.
7	Page change keys	Two kinds of page change keys are described below:  : This key is used to changeover the page on the CRT screen in the forward direction.  : This key is used to changeover the page on the CRT screen in the reverse direction.
8	Program edit keys (ALTER INSRT DELET)	Press these keys when editing the program.
9	Store key	Save data in the CNC memory to the non-voltile memory.
10	Override keys	These keys are used to adjust the override of Spindle speed , Feedrate and Rapid traverse rate respectively.
11	SHIFT	Some keys have two characters on their keytop. Pressing the <SHIFT> key switches the characters.
12	Cycle start key	This key is used to actuate Memory operation or MDI operation .

13	Feed axis and direction selection keys	In the Jog mode,these keys are used to continuously moves the tool along the selected axis in the selected direction. In the STEP mode, pressing these keys to move the tool one step along the selected axis in the selected direction.
14	Rapid move button in JOG mode	This key is used to move the tool at rapid traverse rate in JOG mode.
15	Spindle positive revolution button	This button is used to actuate the Spindle positive revolution.
16	Tool selection manually	Refer the machine manual for detail.
17	Coolant key	Refer the machine manual for detail.

2.6 The keys for manual miscellaneous function

2.6.1 The keys to adjust feedrate override

In JOG mode , the jog feedrate can be adjusted by these keys.

In AUto mode ,the feedrate can be adjusted by these keys.

2.6.2 The keys to adjust the rapid traverse rate override

The following overrides can be applied to a rapid traverse rate with the keys :F0, 25, 50, 100%

F0: Allows a fixed feedrate to be set by parameter P26.

The rapid traverse rate override is effective in the condition below.

<1> The command G00 is executed.

<2> The rapid traverse during the cycles.

<3> The rapid traverse during G27、G28 and G29 commands.

<4> The rapid traverse in JOG mode.

<5> The rapid traverse during manual reference position return.

For example: The rapid traverse rate is 6m/min, the override is 50%, and then the actual speed is 3m/min.

2.6.3 The keys to select the step increment

These keys are used to select the distance to be moved for each step in STEP/HANDLE mode.

2.6.4 The keys to adjust the spindle speed override

These keys are used to specify the spindle speed override.The value range is from 50% to 120% and the least step increment is 10%.

2.6.5 The keys for manual miscellaneous operation

Spindle CW: Spindle starts to rotate clockwise if the key is pressed in JOG or STEP/HANDLE mode.

Spindle CCW : Spindle starts to rotate counter-clockwise if the key is pressed in JOG or STEP/HANDLE mode.

Spindle stop: Spindle stops rotating if the key is pressed in JOG or STEP/HANDLE mode.

Spindle Rotate one step: In JOG or STEP/HANDLE mode,Spindle starts to rotate clockwise if the key is pressed continuously .Spindle will not stop until the key is released.

Coolant switch: Press this key in JOG or STEP/HANDLE mode, it will change ON→OFF→ON… like a lock pushbutton.

Manual Tool selection: Press this key in JOG or STEP/HANDLE mode, tool carrier will rotate and select the next cutting tool. (See the description of machine tool builder for details.)

NOTE1: In JOG or STEP/HANDLE mode ,Pressing any key about Spindle during Spindle rotating will stop it.

In AUTO mode, pressing any key during Spindle rotating to specify a different rotating drection will stop the execution of current program.Furthermore an alarm is output.

2.7 Storage into Disk

Storage key is [save].

2.8 Other button

2.8.1 Cycle start switch: Automatic operation starts.

2.8.2 Three position switch:

Left: normal Middle: feed hold right: spindle and feed hold.

2.8.3 Emergency stop button

2.8.4 System power on button

2.8.5 System power off button

3 MANUAL OPERATION

3.1 Manual Reference Position Return

Press the reference position return switch to select manual reference point return mode. Then press the movable axis selection switch continuously until the tool returns to the reference position.During the tool is in the reference position, the coordinate display on position screen blinks.

NOTE1: When the parameter ZNIK is set 1, the tool can return to the reference position automatically after the switch is pressed once. The tool motion can not be intermitted unless the RESET key is pressed.

NOTE2:The parameter MZRZ、MZRX are used to specify the switch for Manual Reference Position Return.

3.2 JOG FEED

In the jog mode, pressing a feed axis and direction selection switch on the machine operator's panel continuously moves the tool along the selected axis in the selected direction. If the rapid feed switch is pressed simultaneously, the tool will move at the rapid traverse rate. The jog feedrate can be selected by Feedrate Override switches on position screen. (0～1260mm/min)

NOTE1: 1 Feed rate, time constant and method of automatic acceleration/deceleration for manual rapid traverse are the same as G00 in programmed command.

3.3 STEP FEED

In STEP mode, pressing a feed axis and direction selection switch on the machine operator's panel moves the tool one step along the selected axis in the selected direction. The minimum distance the tool is moved is the least input increment. Each step increment can be specified 10, 100, or 1000 times the least input increment by the MPG INC/DEC switches.The last two step increment can be shielded by parameter SINC/P01.

3.4 MANUAL HANDLE FEED

In the HANDLE mode, the tool can be minutely moved by rotating the manual pulse generator on the machine operator's panel. Select the axis along which the tool is to be moved with the handle feed axis selection switches. The minimum distance the tool is moved when the manual pulse generator is rotated by one graduation is specified with the MPG INC/DEC switches as below: 0.001、0.01、0.1mm. The last one can be shielded by parameter SINC/P01.

3.5 MANUAL PROGRAM RETURN ZERO MODE

In manual program return zero mode, operation is the same as manual reference position return. Move to the position specified by G50 at rapid traverse.

1 program zero store:

In program, the position specified by the first G50 command is stored. If another G50

is specified, the position will not be stored.

2 Program zero isn't modified until a new program zero is specified. That is to say, program zero A is specified in program A, when executing program B, program zero A won't change if there isn't G50 command in program B.

EXPLANATION:

After stopped at the middle of the program, the tool can move to machine start point rapidly in manual mode with this function. The tool compensation values will be cancelled automatically.

If program zero isn't stored, alarm No.90 will occur during zero return.

Press address P and key REFERENCE POSITION RETURN simultaneously, program zero return mode will be specified.

3.6 MANUAL AUXILIARY OPERATION

3.6.1 MANUAL TOOL SELECTION

Press this key in JOG or STEP/HANDLE mode, tool carrier will rotate and select the next cutting tool. (Refer to manual of machine tool manufacturer).

3.6.2 Coolant switch

Press this key in JOG or STEP/HANDLE mode, it will change ON→OFF→ON… like a lock pushbutton..

3.6.3 Spindle CW

Spindle starts to rotate clockwise if the key is pressed in JOG or STEP/HANDLE mode.

3.6.4 Spindle CCW

Spindle starts to rotate counter-clockwise if the key is pressed in JOG or STEP/HANDLE mode.

3.6.5 Spindle stop

Spindle stops rotating if the key is pressed in JOG or STEP/HANDLE mode.

3.6.6 Spindle Rotate one step

In JOG or STEP/HANDLE mode,Spindle starts to rotate clockwise if the key is pressed continuously .Spindle will not stop until the key is released.

3.6.7 Setting speed override

4 AUTOMATIC OPERATION

4.1 Operation Mode

4.1.1 Memory Operation

(1) In AUTO mode, select the executed program, and then press the cycle start button. Programs are registered in memory in advance. When one of these programs is selected and the cycle start switch on the machine operator's panel is pressed, automatic operation starts, and the cycle start LED goes on.

(2) Operating from specified program block.

In AUTO mode, Memory Operation can start from a specifed program block.First the program block may be selected by Program search,then the cycle start switch is pressed and automatic operation starts.

NOTE: Before automatic operation starts, Spindle and Coollant can be switched on manually.

4.1.2 MDI OPERATION

In the MDI mode, a program can be inputted in the same format as normal programs and executed from the MDI panel.MDI operation is used for simple test operations. The following procedure is given as an example. For actual operation,refer to the manual supplied by the machine tool builder.

In MDI mode, a program block can be input from LCD/MDI panel, the program can be executed. But the input block can't be stored into system memory.

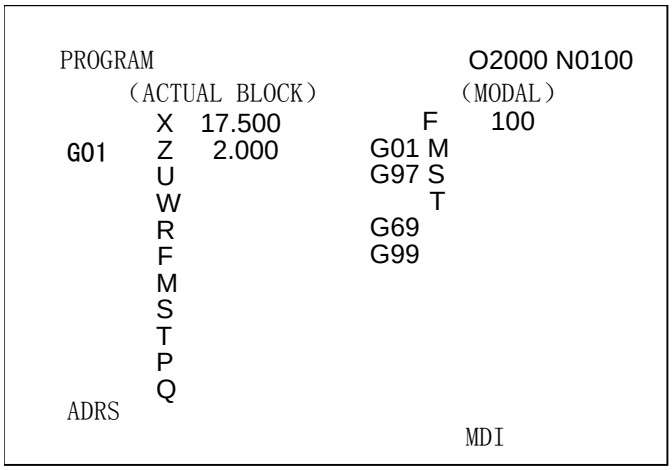
(1) Example: G01 X17.5 Z2. F100;

1) Press MDI key on the mode select switch.

2) Press the function key 【PRGRM】to display program.

PROGRAM		O2000 N0100	
（ ACTUAL BLOCK）		（MODAL）	
X		F 100	
Z	G01	M	
U	G97	S	
W		T	
R	G69		
F	G99		
M	G21		
S		SRPM	0000
T		SSPM	0000
P		SMAX	9999
Q		SACT	0000
ADRS			
	MDI		

3) Input “G01” by address/numeric key.
4) Press INSRT key. The data, G01, is input and displayed. If you are aware of an error in the keyed-in number before pressing the INSRT key , press the CAN key and key in the correct address and number again.
5) Input “X17.5” by address/numeric key.
6) Press INSRT key. The data, X17.5, is input and displayed. If you are aware of an error in the keyed-in number before pressing the INSRT key , press the CAN key and key in X and the correct number again.
7) Input “Z2.” by address/numeric key.
8) Press INSRT key. The data, Z2. , is input and displayed. If you are aware of an error in the keyed-in number before pressing the INSRT key , press the CAN key and key in X and the correct number again.
9) Input “F100” by address/numeric key.
10) Press INSRT key. The data, F100 , is input and displayed. If you are aware of an error in the keyed-in number before pressing the INSRT key , press the CAN key and key in X and the correct number again.



11) Press the cycle start button on the machine operator’s panel.

4.2 STOPPING AND TERMINATING AUTOMATIC OPERATION

Automatic operation can be stopped using one of two methods: Specify a stop command, or press a key on the machine operator’s panel.

4.2.1 PROGRAM PAUSE(M00)

Automatic operation is stopped after a block containing M00 is executed. When the program is stopped, all existing modal information remains unchanged. The automatic operation can be restarted by actuating the cycle operation.

4.2.2 PROGRAM END (M30)

(1) Main program end.
(2) Memory operation is terminated and the reset state is entered.
(3) Contol returns to the start of the program

4.2.3 PROGRAM END (M02)

1) Except remaining the output of M codes unchanged, others are the same as M30.

2) This command must be specified in a sigle block

4.2.4 Feed hold

When Feed Hold button on the operator's panel is pressed during memory operation, the tool decelerates to a stop. Machine tool state is as follows after pressing FEED HOLD button.

1) The feed is stopped after deceleration if the machine tool is moving.
2) Dwell is not continued even in a feed hold state if the dwell is being executed.

4.2.5 RESET

Automatic operation can be stopped and the system can be made to the reset state by using RESET key on the LCD/MDI panel or external reset signal. When reset operation is applied to the system during a tool moving status, the motion is slowed down then stops.

5 TEST OPERATION

The following functions are used to check before actual machining whether the machine operates as specified by the created program.

Press the [OPR/GRAPH] key to select TEST OPERATION.

5.1 In/out chuck select(KEY 0)

It is used to select different chuck, and is effective after power off.

5.2 DRY RUN(KEY 1)

Press key 1 to set the DRY RUN on during TEST OPERATION .When the DRY RUN is on , the tool does not move but the position along each axis changes on the display as if the tool were moving. But auxiliary function M, S, and T are unavailable. This is available for checking a program.

NOTE1: When DRY RUN or SINGLE BLOCK is on,the “TEST” will be blinking on the display screen.

5.3 SINGLE BLOCK (KEY 2)

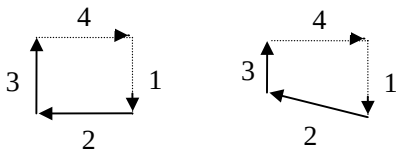
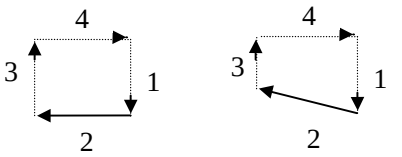
Press key 2 to set the SINGLE BLOCK on during TEST OPERATION .

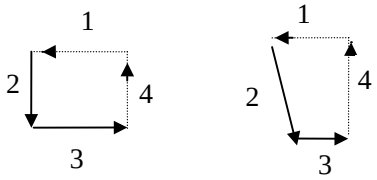
When the cycle start button is pressed in the single block mode, the tool stops after a single block in the program is executed.

NOTE1: If G28 is issued, the single block function is effective at the intermediate point.

NOTE2: In a canned cycle, the single block stop points are as follows：

(Rapid traverse ——— Cutting feed)

G code	Tool track	Explanation
G90		Tool path 1 to 4 is assumed as a cycle. After 4 is finished, a stop is made.
G92		Tool path 1 to 4 is assumed as a cycle. After 4 is finished, a stop is made.

G94		Tool path 1 to 4 is assumed as a cycle. After 4 is finished, a stop is made.

NOTE3: Single block stop is not performed in a block containing M98P_;. or M99.However, single block stop is even performed in a block with M98P_ command, if the block contains an address other than N or P.

5.4 parameter or program switch (key 3,key 4)

<1>parameter protection switch (key 3): when the switch is on,the parameter can be edited.

<2>program protection switch (key 4): when the switch is on，the program can be edited.

5.5 OUTPUTS OF MANUAL ASSISTANT FUNCTIONS

In one of JOG,STEP,HANDLE or REFERENCE POSITION RETURN, pressing key 5～9 can control AUXILIARY OPERATION.

<1> Spindle CW (5 key)

<2> Spindle stop (6 key)

<3> Spindle CCW (7 key)

<4> Coolant switch (8 key)

<5> Lubrication switch (9 key)

6. SAFTY OPERATION

6.1 EMERGENCY STOP

If you press Emergency Stop button on the machine operator's panel, the machine movement stops in a moment. All output such as spindle rotate and coolant will be closed. All output must restart after releasing the button.

This button is locked when it is pressed. The method of release is turning clockwise with pushing.

Red

EMERGENCY STOP

NOTE:

1 EMERGENCY STOP interrupts the current to the motor.

2 Causes of trouble must be removed before the button is released.

6.2 OVERTRAVEL (SOFT LIMIT)

When the tool tries to move beyond the stored stroke limit set by the parameter, an alarm occurs. the tool decelerates and stops because of working the limit and an OVERTRAVEL is displayed. Press the reset button to reset the alarm after moving the tool to the safety direction by manual operation. For details on operation, refer to the operator.s manual of the machine tool builder.

7.ALARM FUNCTION

When a unusual operation occurs, affirm the following contents:

an alarm is displayed on the LCD

Refer to APPENDIX 4—Alarm list to find the cause of the alarm. The causes of alarms are classified by error codes. If PS □□□ is displayed, it is error about program or data setting, an alarm can be cleared by modified program or setting data.

without an alarm on the LCD

Make sure that the system position and executing content by the display on LCD screen. Refer to APPENDIX 3— Diagnose status to find the cause of the alarm.

8 PROGRAM STORAGE AND EDITING

8.1 PREPARATION

(1) Set program protection switch to ON.
(2) Select EDIT mode.
(3) Press the PRGRM to display program.
If communicating with computer, the following is necessary.
(1) Connect PC computer and run KND communication software.
(2) Set the content with respect to RS232. (see 10.2)

NOTE:

A program protection switch is set on 【OPR/GRAPH】soft operator's panel in order to protect machining program. Editing program is not available until the switch is turned on. Pressing display function keys 【OPR/ GRAPH】to display machine tool screen, then press key 【4】to turn on or off program protection switch.

8.2 PROGRAM STORAGE TO MEMORY

(A) Set the mode selection switch to EDIT.
(B) Press the [PRGRAM] button.
(C) Press the address key O.
(D) Press program number.
(E) Press [INSRT] button.

The above procedure is to store program number. Every word in the program is input by repeating the above procedure.

8.3 PROGRAM SEARCH

When memory holds multiple programs, a program can be searched for. In general, the program pointed by current program pointer is displayed. The program pointer can't lose even if turning off the power. The desired program can be called by searching (the program pointer will be changed at the same time), the called program can be edited or execute. This operation is called program search.

There are two methods as follows.

(1) Procedure for searching
(A) Mode selection.(EDIT or AUTO)
(B) Press the [PRGRM] button to display program.
(C) Press the address key O.
(D) Input the desired program number.
(E) Press the CURSOR ↓key.
(F) The searched program displays on the screen and the searched program number appears at the right of the screen after searching.

(2) Procedure for scanning
(A) Mode selection.(EDIT or AUTO)
(B) Press the [PRGRM] button to display program.
(C) Press the address key O.

(D)Press the CURSOR ↓key. In EDIT mode, press the address key O and CURSOR ↓key to display the registered program one by one.

NOTE:

When all registered programs are displayed, the first program will appear.

8.4 DELETING PROGRAM

Delete the program registered in the memory.

(1) Set the mode selector switch to EDIT.
(2) Press the [PRGRM] button to display program.
(3) Press the address key O.
(4) Input the desired program number.
(5) Press the key [DELET] to delete the program corresponding to the input program number.

8.5 ALL PROGRAMS DELETING

Delete all programs registered in the memory.

(1) Set the mode selector switch to EDIT.
(2) Press the [PRGRM] button to display program.
(3) Press the address key O.
(4) Press the keys –9999 and [DELET] key.

8.6 SEQUENCE NUMBER SEARCH

Sequence number search operation 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 CNC. This means that the data in the skipped blocks such as coordinates and M, S, and T codes does not alter the CNC coordinates and modal values. So, in the first block where execution is to be started or restarted by using a sequence number search command, be sure to enter required M, S, and T codes and coordinates. A block searched for by sequence number search usually represents a point of shifting from one process to another. When a block in the middle of a process must be searched for to restart execution at the block, specify 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.

(1) Procedure for sequence number search

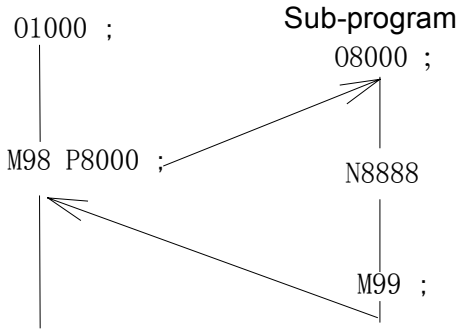
(A) Set the mode selector switch to AUTO or EDIT.
(B) Press the key [PRGRM] to display the program.
(C) Select the program including the desired sequence number.
(D) Press the address key N.
(E) Input the searched sequence number.
(F) Press the key CURSOR↓.
(G) The searched sequence number displayed at the top of the right LCD screen.

NOTE:

1 During search operation, the following checks are made :

- Optional block skip
- P/S alarm (No. 003 to 010)

2 During sequence number search operation, M98Pxxxx (subprogram call) is not executed. So an alarm (P/S No. 060) is raised if an attempt is made to search for a sequence number in a subprogram called by the program currently selected.



If an attempt is made to search for N8888 in the example above, an alarm is raised.

8.7 INSERTING, ALTERING AND DELETING A WORD

The content registered in memory can be modified.

(1) Set the mode selector switch to EDIT;
(2) Press soft key [PRGRM] to display the program.
(3) Select the desired program;
(4) Search for a word to be modified.
- Scan method
- Word search method

(5) Perform an operation such as altering, inserting, or deleting a word.

8.7.1 WORD SEARCH

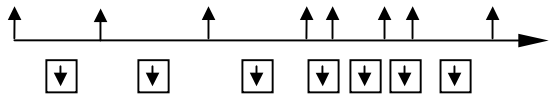
A word can be searched for by merely moving the cursor through the text (scanning), by word search, or by address search.

(1) Word scan

The cursor moves forward word by word on the screen; the cursor is displayed below the address character of a selected word.

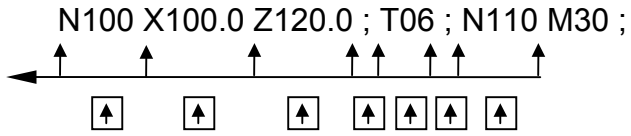
(A) Press the CURSOR↓ button

N100 X100.0 Z120.0 ; T06 ; N110 M30 ;



The cursor moves forward word by word on the screen; the cursor is displayed at a selected word. The cursor is positioned to the address of the selected word.

(B) Press the CURSOR↑ button



The cursor moves backward word by word on the screen; the cursor is displayed at a selected word.

(C) Holding down the cursor key↓ or ↑ scans words continuously.

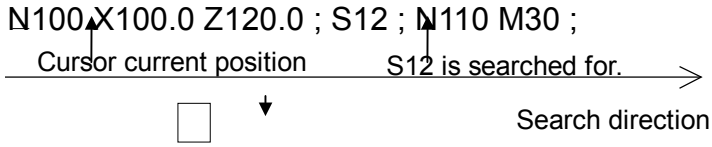
(D) Pressing the page key↓displays the next page and searches for the first word of the page.

(E) Pressing the page key↑displays the previous page and searches for the first word of the page.

(F) Holding down the page key↓or↑ displays one page after another.

(2) Word search

A specified words is searched from the current position in the forward or backward direction.



(A) Key in address S .

(B) Key in ‘1’ ‘2’.

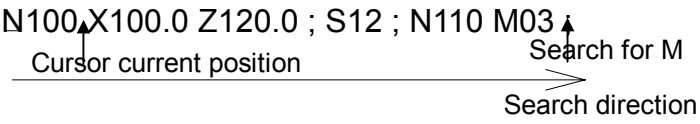
NOTE:

- S12 cannot be searched for if only S1 is keyed in.
- S09 cannot be searched for by keying in only S9. To search for S09, be sure to key in S09.

(C) Pressing the cursor ↓key starts search operation. Upon completion of search operation, the cursor is displayed at “S” of S12. Pressing the cursor key↑ rather than the cursor key↓ performs search operation in the reverse direction.

(3) Procedure for searching an address

A specified address is searched for from the current position in the forward direction.

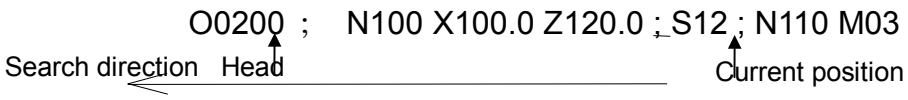


(A) Key in address M .

(B) Press the cursor key↓ .

Upon completion of search operation, the cursor is displayed at “M” of M03. Pressing the key↑ rather than the key↓ performs search operation in the reverse direction.

(4) Heading a program



(A) 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.

(B) Method 2

Execute a Program Number Search.

(C) Method 3

(1) Set the mode selector switch to AUTO or EDIT.
(2) Press the key [PRGRM] to display the program.
(3) Press the address key O
(4) Press CUROSR↑key

8.7.2 Inserting a Word

1 Search for or scan the word immediately before a word to be inserted.
2 Key in an address to be inserted.
3 Key in data
4 Press the INSRT key.

8.7.3 Altering a Word

(1) Search for or scan a word to be altered.
(2) Key in an address to be inserted.
(3) Key in data.
(4) Press the ALTER key. The input word will replace the word pointed by current cursor.

8.7.4 Deleting a word

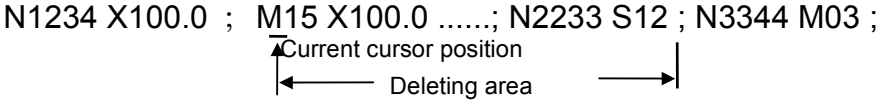
(1) Search for or scan the word to be deleted.
(2) Press the [DELET] button to delete the content pointed by the current cursor.

8.7.5 Deleting up to an EOB(；)

To delete area from the current cursor position to EOB, operation is as follows:
1) Move the cursor to the next word address
2) Press the [EOB] key and the [DELET] button.

8.7.6 Deleting multiple Blocks

Delete the blocks from the current display to the specified sequence number block.



(1) Press the address key N

(2) Input the sequence number 2233
(3) Press the [DELET] button to delete up to block N2233 and the cursor move to the next address word.

8.8 Memory Capacity

Total 63 program can register in the memory simultaneously.

8.9 Storage Capacity

(1) Program storage capacity
Memory: 80m (4KB = 10m).
External Memory：flash memory（6-memory）
(2) Compensation Data: 8

9 DATA DISPLAY AND SET

9.1 Setting and Display of the tool compensation【OFSET 】

Display and setting of the tool compensation values have two methods: absolute input and incremental input.

(1) In absolute mode

1) Press the〖OFSET〗button.
2) Select the desired page by pressing the PAGE button.
3) Move cursor to the offset number to be changed.
4) Input address X or Y and the offset values (including decimal) using data keys.
5) Press the key INSRT and display the offset amount on the LCD screen.

(2) In incremental mode

1) Move cursor to the offset number to be changed. (see the above)
2) Input address U or W and incremental values(including decimal) using data keys.
3) Press the key INSRT and add incremental values to current offset and display.

NOTE:

1 When the offset value is changed during automatic operation, the new offset value does not become valid until the H code followed by the offset number is commanded.

2 The compensation and offset are the same meaning in this description. That is to say, compensation number is equal to offset number and compensation values is equal to offset amount.

9.2 PARAMETER

When connecting CNC and machine tool, set parameter to make the function of driver character and machine tool specification well. For details refer to the description of individual machine tool builder.

9.2.1 Parameter Display

1) Press the button PARAM
2) Press the page key to select display screen.

On the parameter screen, there is details for parameter at the bottom of the LCD screen.

(1) Bit parameter

Parameter №001 to 004 and №041 to 044 is bit parameter, high bit is on the left (BIT7), the sequence is BIT6，BIT5，BIT4 and BIT3，BIT2，BIT1，BIT0. So far as concerns bit parameter, English abbreviation for the overall parameter bit, is displayed.

(2) Data parameter

The display method of parameter details is only one kind. For example, when the cursor is at the position of No.005, the display is as follows:

X-axis command multiplication rate.

9.2.2 Set Parameter

Parameter can be set on the MDI/LCD panel.

1) Set parameter switch to on.
2) Select MDI mode (or press the emergency button)
3) Press the key PARAM to display the parameter screen.
4) Press the page key to display the parameter to be set.
5) Move the cursor to the parameter number to be set.
6) Press the number key to input parameter value.
7) Press the key INSRT to input and display the parameter.
8) After the completion of the overall parameter setting, select the set screen and set parameter switch to off.
9) Press the key RESET to reset the alarm. But if alarm No. 000 occurs, it is necessary to turn on or off the power in order to reset the alarm.

9.3 Diagnose

The diagnose function can display the following signal status.

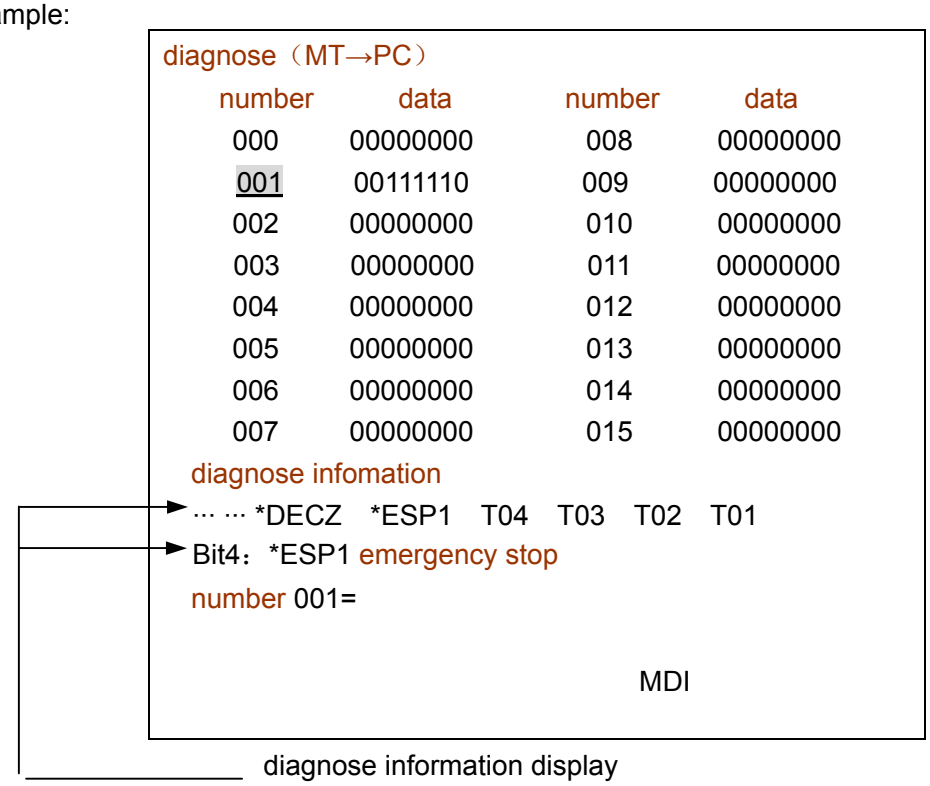
- The status of DI/DO signal between CNC and machine tool
- The signal status transmitted between CNC and PC
- The data inside PC
- The CNC inner status

The parameter can output to machine tool directly according to corresponding set. Auxiliary function parameter can be set in this mode.

Diagnose Display

1) Press the button DGNOS
2) Move the cursor to select the desired diagnose number.
3) Prsee INSRT and key 1 to display the details of the selected diagnose number. There are three display-bars at the bottom of the LCD screen to display the details of the diagnose number with the cursor.
4) Press CAN to cancel the details and other diagnose numbers can be selected.

Example:



Press diagnose key to enter diagnose page. Move cursor to number 001,check the last four bits,if the first bit of number 001 changes to 0, that is 00111110, which indicate current tool is No 1. The abnormal pars in the machine can be check out quickly , and it is convenient for repair.

10 DISPLAY

10.1 Status Indication

The status of the system is indicted at the bottom of the screen. The following indications are displayed.

NOT READY	Indicates that the control unit or the servo system is not ready for operations and this indication is flickering.
ALM	Indicates that an alarm was generated, press the ALARM button to display alarm details (this indication is flickering).
BAT ALM	This alarm displays when the voltage of the battery is less than the specified value.
OPR	Display the current operation mode: AUTO, EDIT, JOG, ZRN, STEP/HNDL, MDI, RETURN PROGRAM ZERO.
EDIT	Indicates that the current editing operation. A) EDIT: Indicates that editing is being performed. B) SEARCH: Indicates that searching is being performed. C) SAVE: Indicates that system data is being stored to flash memory.

10.2 KEY IN DATA DISPLAYING

The inputting values displays in the line above the status indication.

The symbol for prompting: it is existing only on the screen which can be input.

A) Display program in EDIT mode:

ADRS	Input address only
DATA	Input data only

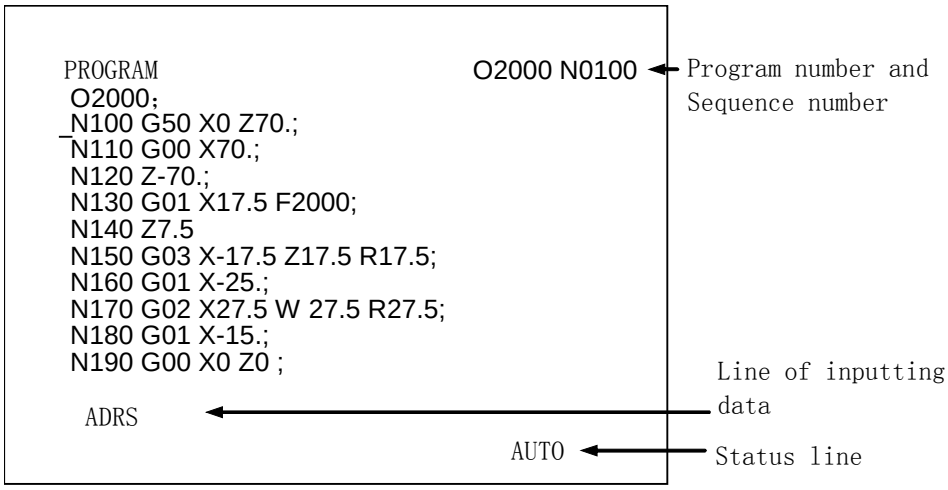
B) PARAM, OFSET, AGNOS screen:

No.005=…	Input parameter values available.
No.005…	The input data are invalid.
No.005 flickering	Key in the searched sequence number.

Display the input data behind the symbol for prompting and press the button INSRT , the input values disappears.

10.3 Displaying the Sequence Number and Program Number

The program number and sequence number are displayed at the top right on the screen as shown below.



The program number and sequence number displayed depend on the screen and are given below:

On the program screen in the EDIT mode on Background edit screen :

The program No. being edited and the sequence number just prior to the cursor are indicated.

Other than above screens :

The program No. and the sequence No. executed last are indicated.

Immediately after program number search or sequence number search :

Immediately after the program No. search and sequence No. search, the program No. and the sequence No. searched are indicated.

10.4 Display the Program Memory Usuage

The display method is as follows.

1) Mode select. (EDIT)

2) Press the button PRGRM twice to enter the page.

3) The following is displayed.

(A) Registered program number: the number of registered programs(including sub-program)

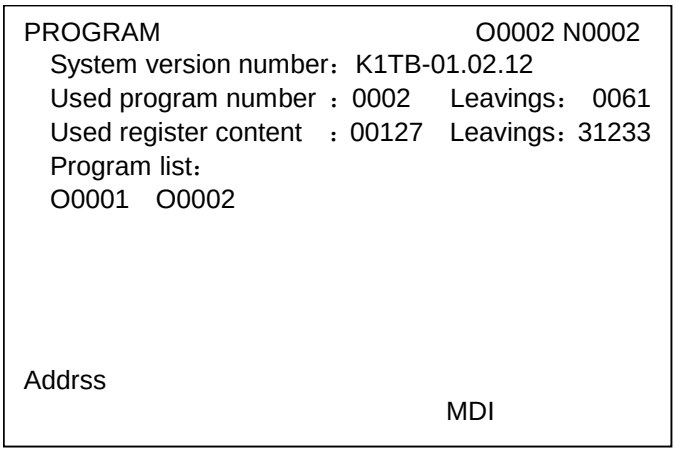
(B) Remainder：the number of programs to be stored

(C) Used memory: memory capacity taken up by registered program(expressed by the number of word characters)

(D) Remainder：the number of program memory capacity to be used

(E) Program list: display the registered program number one by one

Press CAN key to return to program screen.



10.5 Coordinate display and clearing

10.5.1 COORDINATE DISPLAY

(1) Press the button POS

(2) Press the button PAGE to switch to one of the four pages.

There are three coordinates to be displayed.

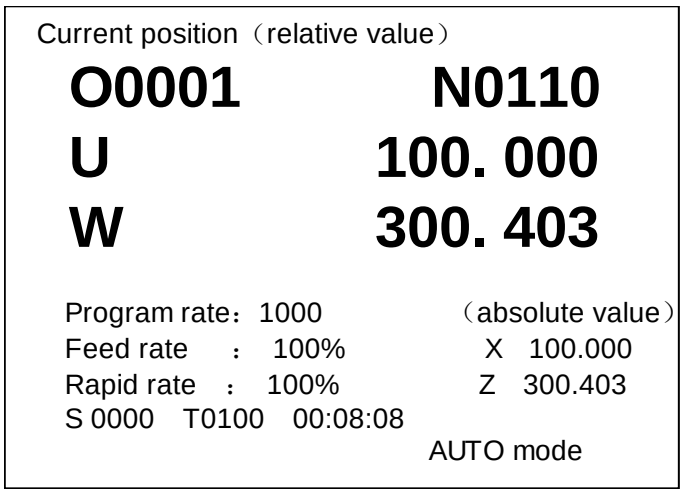
1 Increment coordinate: With address U,W. It can be changed by G50 and can be cleared at any time. It can be used for observing and setting tool offset.

2 Absolute coordinate: With address X,Z. It corresponding to program absolute value and is also be called workpiece coordinate and can be changed by G50.

3 Machine coordinate ： With address X,Z. The reference point of machine tool is coordinate zero. In a general way, it can't be changed and be used for checking soft travel limit.

Select position screen, press PAGE key to display the four screens.

<1> Display increment coordinate screen



NOTE1: The value following address S is euql to the actual spindle rotation rate only if a position decoder with 1024 line has been equiped.

NOTE2: Program rate＝program F rate × Feed rate override. The value can not be

displayed under the following conditions: dry run,G00 or the value of parameter NO.24 is regarded as upper limit.

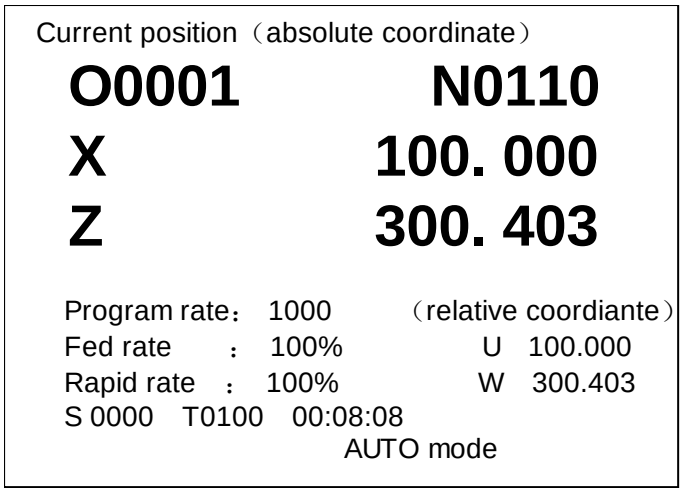
NOTE3：When feed per rotate or thread cutting is effective, the display unit of program rate is 0.001mm/r.The least two digit can't be displayed.

For example: G99 F20.2568 displayed as 2025

G99 F10 displayed as 1000

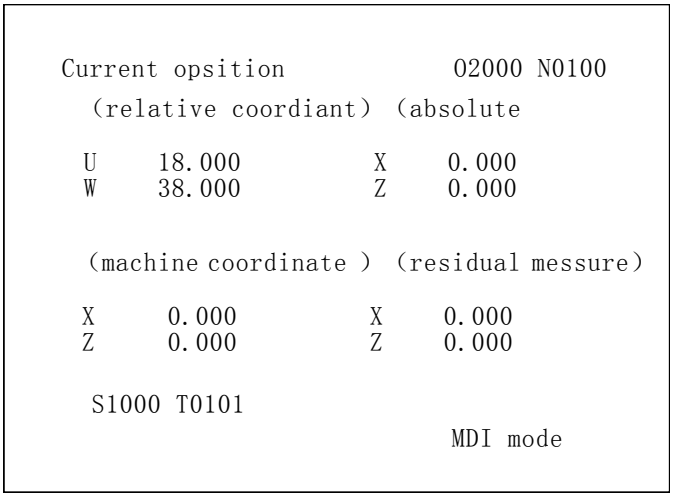
NOTE4: If actual feedrate is over the upper limit, "***" is displayed

<2> Display absolute coordinate screen



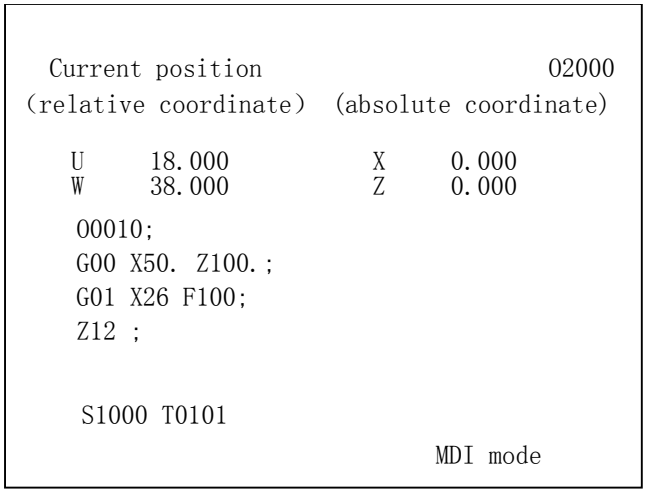
<3> Display all coordinate screen

The third page with small character is all coordinate screen.



<4> Position and program screen

The mixing of position and program is displayed on fourth page.



10.5.2 COORDINATE CLEARING

Procedure to reset the axis increment coordinate to 0:

1 Key in the address of the axis name (U, W,) on the increment coordinate screen. The entered axis address blinks.

2 Press the CAN key. The relative coordinates of the axis having the blinking address are reset to 0.

3 Press the key (U,W) again ,the axis address stop blinking.

Procedure to reset the machine coordinate to 0:

Key in the address of the axis name X or Z with the CAN key being pressed simultaneously on the all coordinate screen. the macnine coordinates of the selected axis is reset to 0.

10.6 Display of Run Time and Parts Count

The run time, cycle time, and the number of machined parts are displayed on the current position display screens.

10.8 Display Alarm

When an alarm occurs, ALM is flickering at the bottom of the LCD screen. Press the function key or soft key ALARM to display the alarm number and content. The meaning of alarm number are list in the APPENDIX.

There is an alarm details display at the bottom of the LCD screen on alarm screen to display the current P/S alarm details. Other alarms such as driver device alarm and overheat alarm display in the middle of LCD screen.

NOTE:

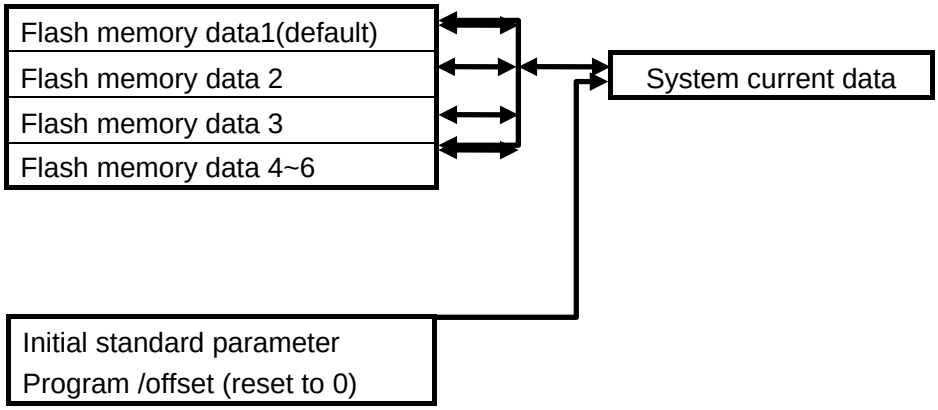
1 Display screen is changed to alarm screen automatically when an alarm occurs.

2 In feed hold mode and without alarm, “DWELL” will be flickering at the bottom of the screen instead of “ALARM”.

11 FLASH MEMORY

11.1 GENERAL

This system adapt flash memory. It is non-volatile memory.



There are eight disks in a flash memory, each disk includes data such as parameter, program, offset, PC parameter and so on. The system current data can be stored in a random disk, in the meantime , system can read data from a random disk as the current data.

There are two kinds of versions system, the difference between them is as follows.

11.2 Read from disk

Read data from a random disk to the working area, the procedure is as below:

1 Press INSRT + 0~6 number key and turn on the power simultaneously

2 The following words are appeared:
insert a disk, press RESET to conform or press CAN to cancel (number)

3 Press the key RESET, the disk data corresponding to the number key are read into the working area. Press the key 0 to set KND initial standard parameter, and the data in program area and offset area will be reset to 0, in order to the first install of new flash memory.

NOTE:

1 The default value is disk 1 without pressing power-on in version 1.

2 Read is being executed only in power-on, read can't be executed after turning on the power.

11.3 Storage into Disk

The procedure is as follows.

1 Display the program screen.

2 Mode select.(EDIT)

3 Press the address key N and number key 1~8 and the button STORE.

"Save" is flickering at right-down of the status indication during storage. The default value is disk1 when the number key is ignored. The flickering "save" will disappear after the completion of the storage.

After inputting parameter and program data, “please store” is flickering at right- down of the screen automatically. The modified data must be stored in order to remember after power off. “please store” will disappear automatically after storage.

NOTE:

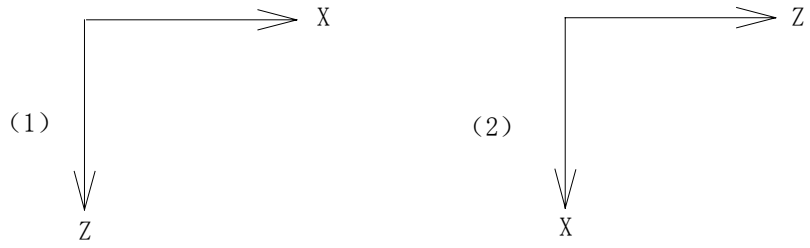
1 During storage, if power-off occurs, the new data can not be stored and the data in the original disk will also lose. Pick out a disk (from 6 disks) as buffer disk during operation, and store to buffer disk first. Thus, the data in the original disk will not lose in power-off, if power-off occurs, the data will be read to buffer disk in power-on and then store.

Storage can be executed when the ESP is on.

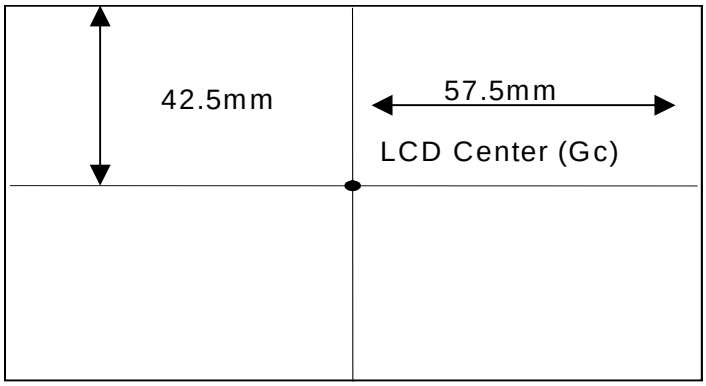
12. GRAPHIC FUNCTION

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.

Using graph parameter to select one of the two coordinate systems:



The dimension of LCD:



The maximum dimension of graph on the LCD screen is 115mmX85MM. If the workpiece shape is large than the maximum graph dimension, the machined graph must be reduced and enlarged. The range of reduce and enlarge proportion is 0.01 to 100.00. The method of reduce and enlarge in 2-dimensional space is as follows.

Scale Proportion = MIN（horizontal proportion，vertical proportion）；

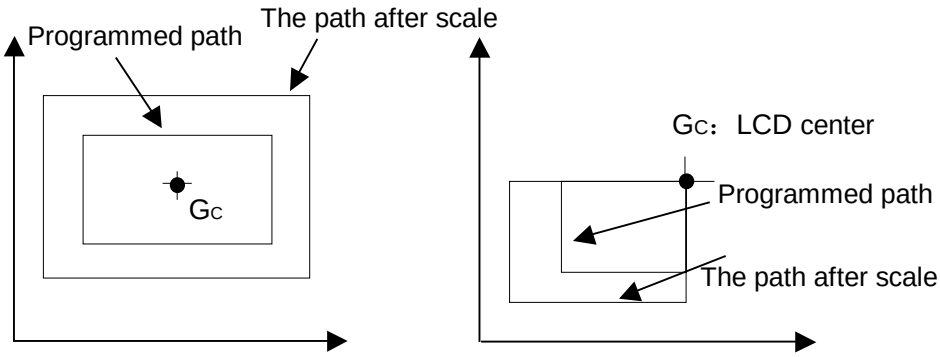
Horizontal proportion = α/machined length in horizon

Vertical proportion = β/machined length in vertical

α ： 115mm

β ： 85 mm

The basic point of scale is at the center of the LCD screen.



The tool path on the LCD screen is described by the workpiece coordinate values of the tool moved. The center of LCD corresponding to the workpiece coordinate values is as below.

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 graph parameter.

12.1 Set Graph Parameter

Graph parameter must be set as showed in Fig. below before start-up.

(1) Press the key GRAPH twice, graph parameter is displayed on the screen.

```
GRAPH                                        O0010 N0010
                    Graph parameter

Coordinate system selection =  0        （XZ0，ZX1）
Scale proportion = 100
Graph center   =  25.000(X-axis workpiece coordinate system）
Graph center   =  30.000(X-axis workpiece coordinate system）
X MAX =            50.000
Z MAX=                  20.000
X MIN =            60.000
Z MIN =            10.000

   No. 001 =
                                        MDI
```

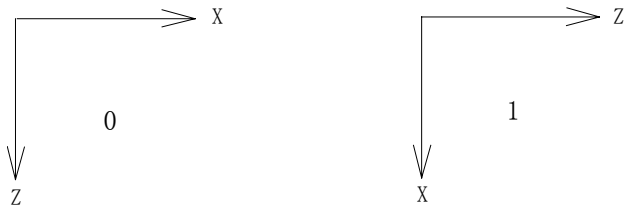
(2) Press the cursor key ↓ , ↑ and move the cursor under the parameter to be set.

(3) Press the number key and the button INPUT. Input graph parameter values.

(4) Repeat the procedure (2) (3) to set parameter to be set.

12.2 DESCRIPTION OF GRAPH PARAMETER

• **Select coordinate system：Coordinate plane selection**（XZ = 0，ZX = 1）



• **Scale proportion：Set drawing proportion.**

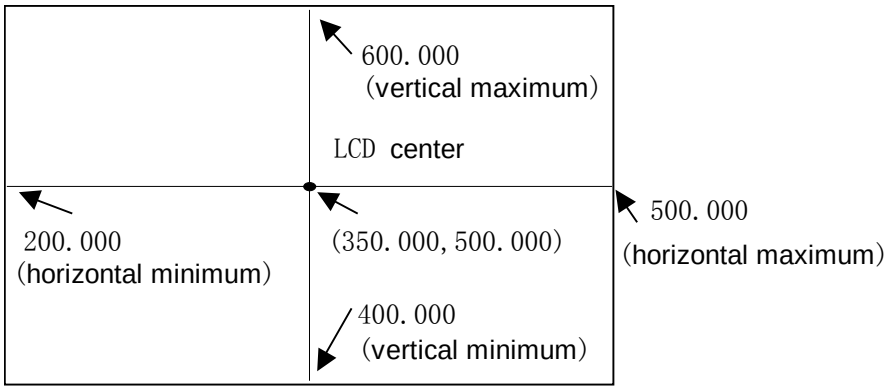
Set range： 1～10000

1 = 0.01 times

• **Graph center：Set the coordinate values of LCD center in workpiece coordinate system.**

• **Maximum and minimum values：Set the maximum and minimum values of each axis, then corresponding graph center coordinate values can be set automatically.**

EXAMPLE:



horizontal graph center =(500.000+ 200.000)÷2 = 350.000

Proportion= 115÷300 = 0.383

vertical graph center =(600.000+ 400.000)÷2 = 500.000

Proportion= 85÷200 = 0.425

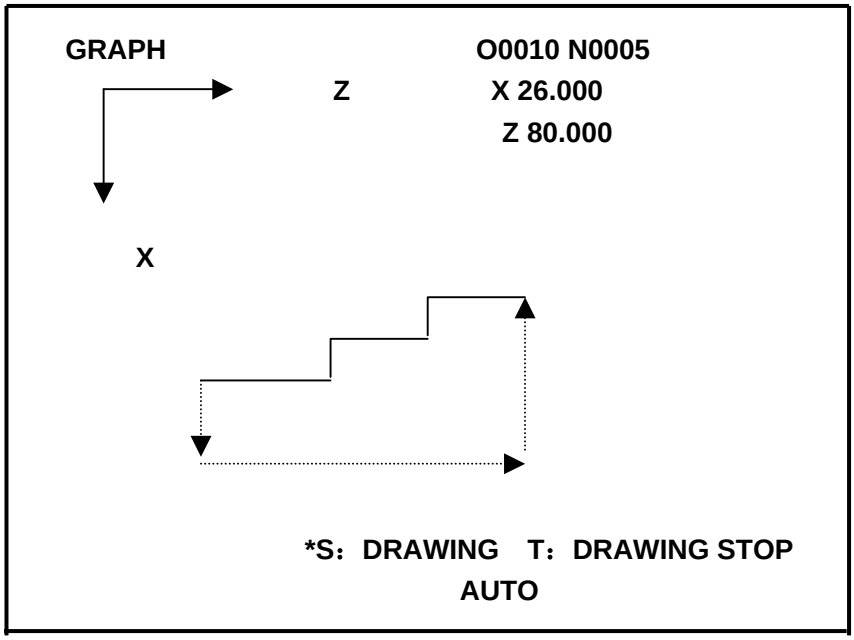
Scale proportion = min｛0.38,0.42｝= 0.38,

It can be set as the values no more than 38.

If graph center parameter need to modify, it can be set after setting the maximum and minimum values of each axis.

12.3 DESCRIPTION OF TOOL PATH

Drawing screen is on the second page of the graph screen, it can be selected by the key PAGE：



① Press the key S to enter drawing mode.
② Move machine tool in AUTO/MDI/JOG mode. When the absolute coordinate values change, tool path will be displayed correspondingly.
③ Press the key T to enter drawing stop mode.
④ Press the key R to delete drawing.

NOTE:

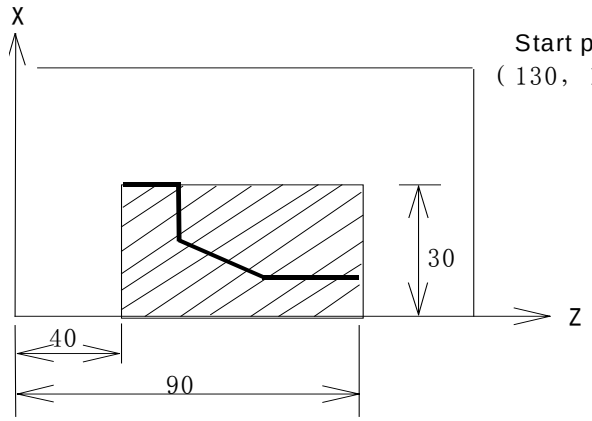
In program debugging, machine lock and dry run feed rate can be used.

12.4 EXAMPLE

Parameter selection:

Programmed unit 0.001mm

Drawing plane 1



(a) Draw overall tool path；

Set the maximum and minimum：

X max = 130000，X min = 0

Z max = 150000，Z min = 0

Graph center is set automatically.（65000，75000）

Scale proportion (horizontal)= 115÷150 = 0.76

Scale proportion (vertical)= 85÷130 = 0.65

Scale proportion coefficient is no more than 0.65 (65)

(b) Draw the thalweg

X max = 30000，X min = 0

Z max = 90000，Z min = 40000

Graph center is set automatically（15000，65000）

Scale proportion (horizontal)= 115÷50 = 2.30

Scale proportion (vertical)= 85÷30 = 2.8

Scale proportion coefficient is no more than 2.3 (230)

(c) Offset on the LCD screen

Change the maximum and minimum with the same data (or change the graph center directly)

max+ a, min+ a

a＜0， drawing position shift upward or towards the right

a＞0， drawing position shift downward or towards the left

3-dimension space：

a＜0， drawing position shift towards the positive end of axis

a＞0， drawing position shift towards the negative end of axis

Ⅳ CONNECTION

1 System Structure

1.1.System Configuration

KND- 1TBⅡ series CNC system is made up of the following units:

(1) CNC control unit(KND -1T BⅡ）
(2) Stepper motor driver device (digital AC servo motor)
(3) Stepper motor (servo motor)
(4) Power transformer
(5) Electrical cabinet

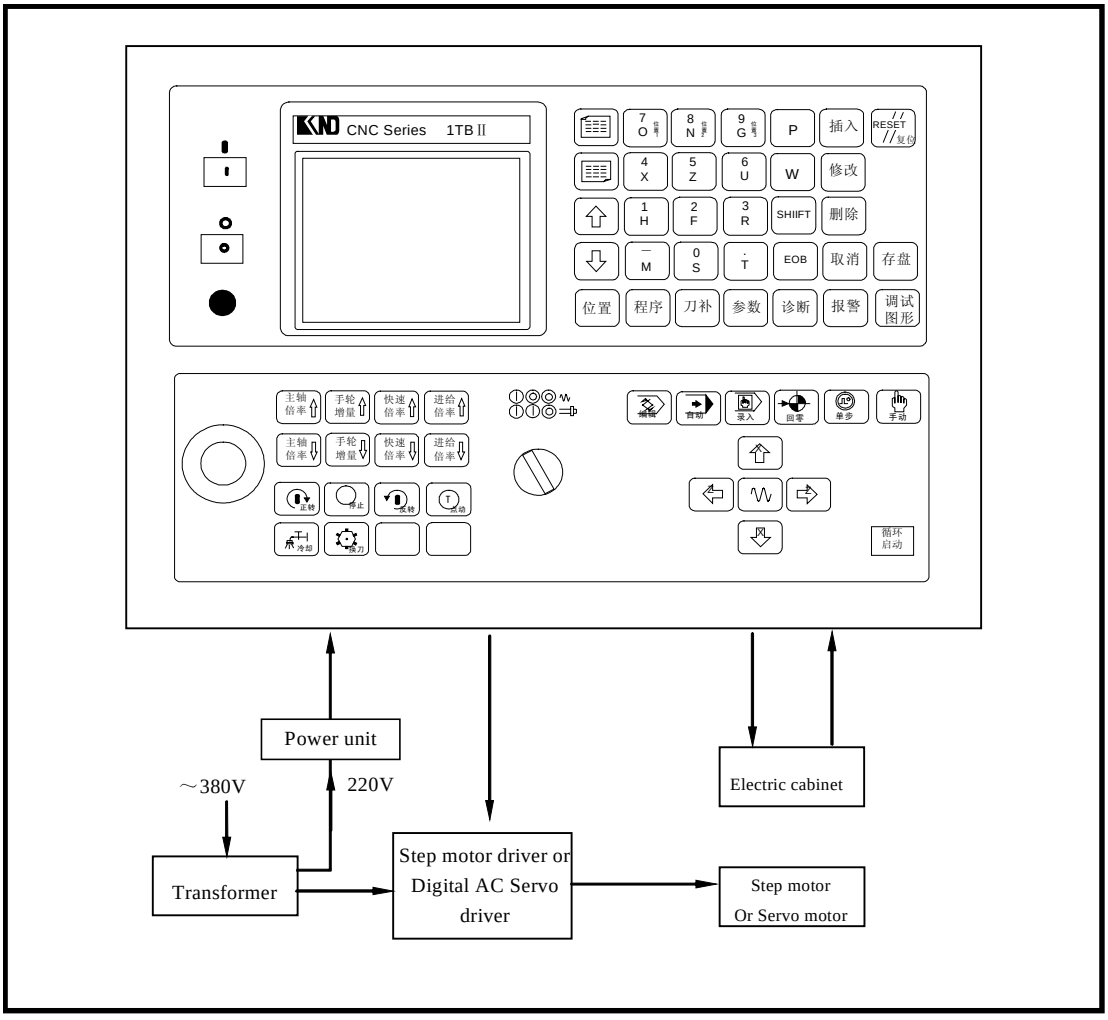


Fig. 1.1

Note：System power capacity （200～240VAC）50W

1.2. Installation Dimension of CNC Control unit

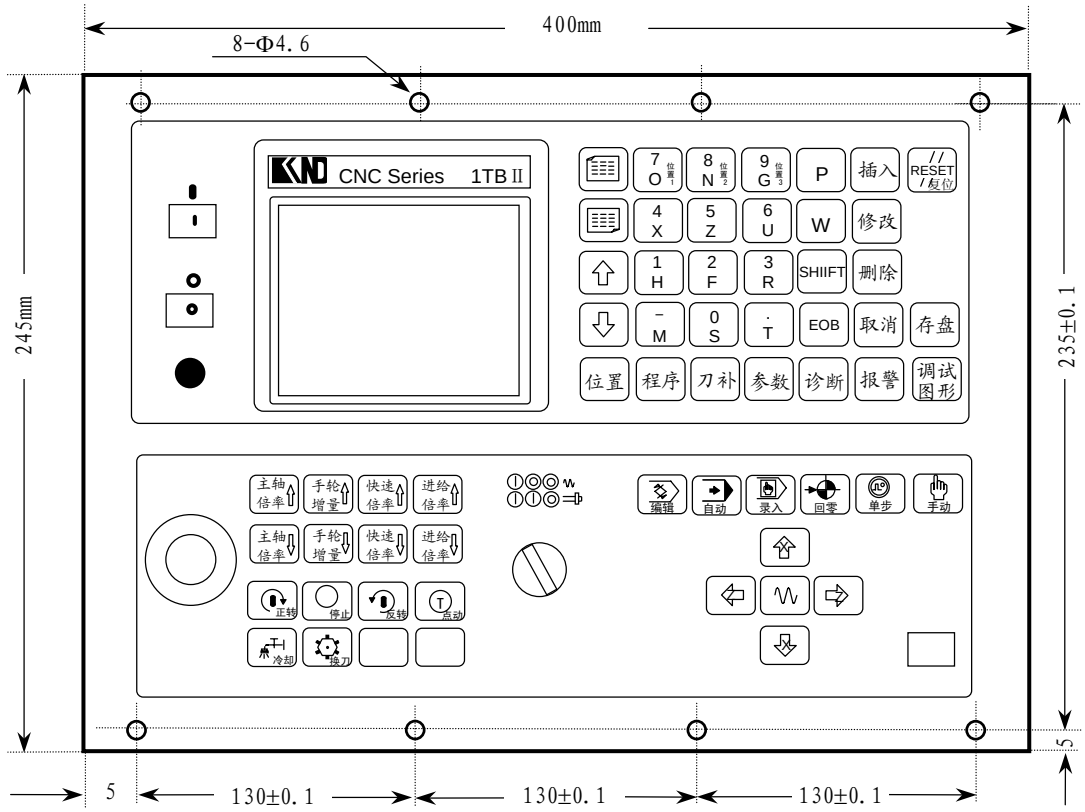


Fig. 1.2a front view

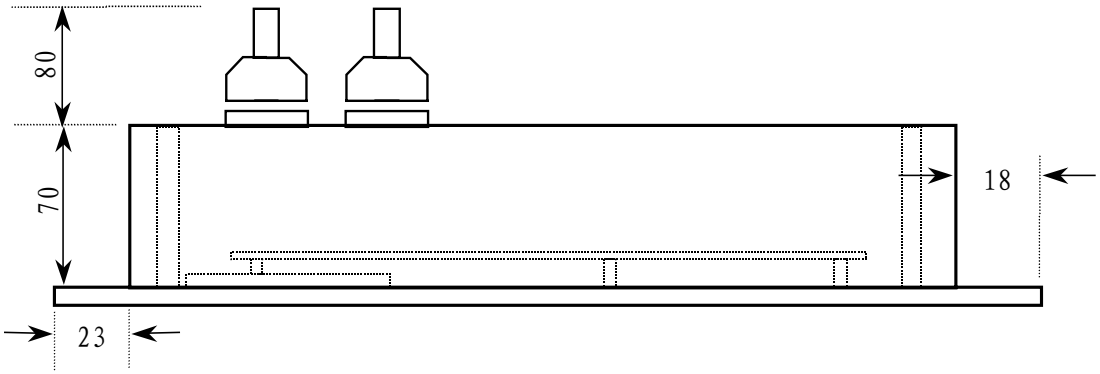


Fig. 1.2b upside view

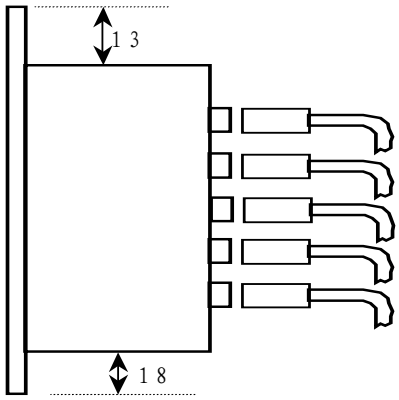


Fig. 1.2c left view

1.3 Additional Operator's Panel Dimension

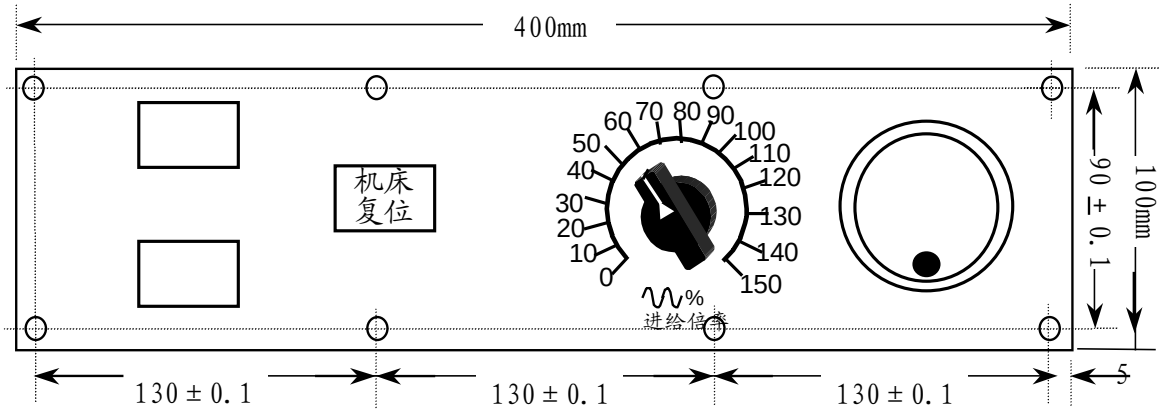


Fig. 1.3

1.4 Power unit D-50B Dimension

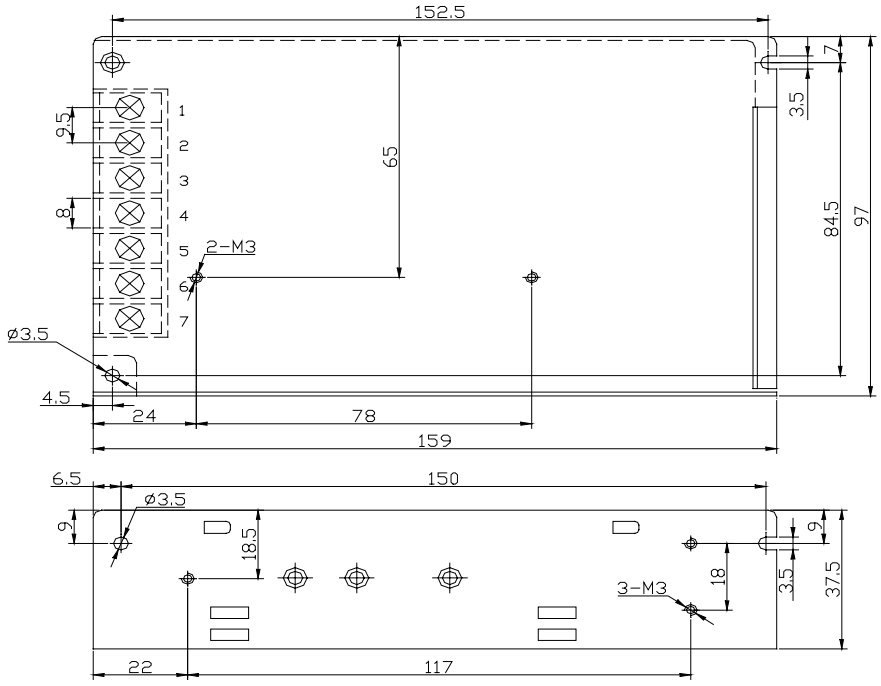


Fig. 1.4

2 INTERNAL CONNECTION

2.1 The PCB of K1TBⅡCNC sytem

2.1.1 Main board（Version：（AB）/08I-W01Z-0108）

Fig. 2.1.1

2.1.2 LCD board of K1TBⅡ

Fig. 2.1.2

2.2 System Internal Connection Diagram

2.2.1 The connection with Main board

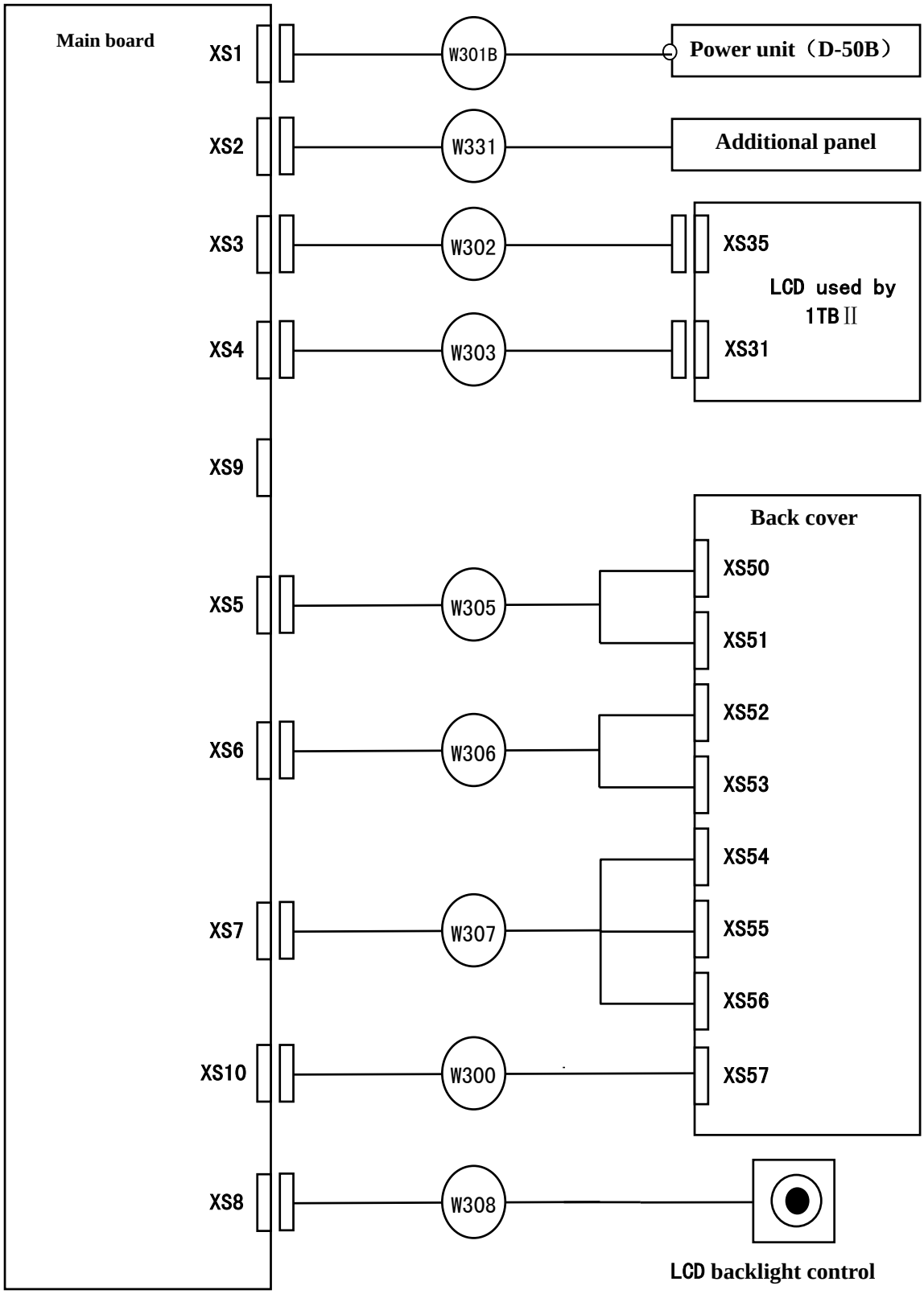


Fig. 2.2.1

2.2.2 Connection between LCD board and LCD screen

(1)OPTREX LCD used by 1TBⅡ

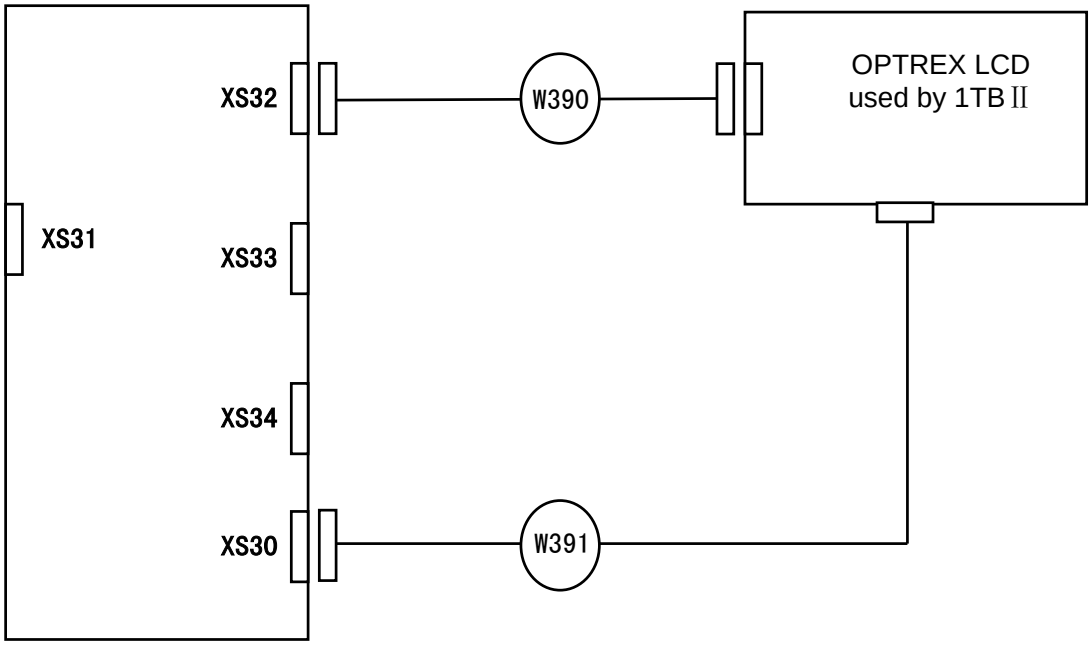


Fig. 2.2.2a

(2) SHARP LCD used by 1TBⅡ

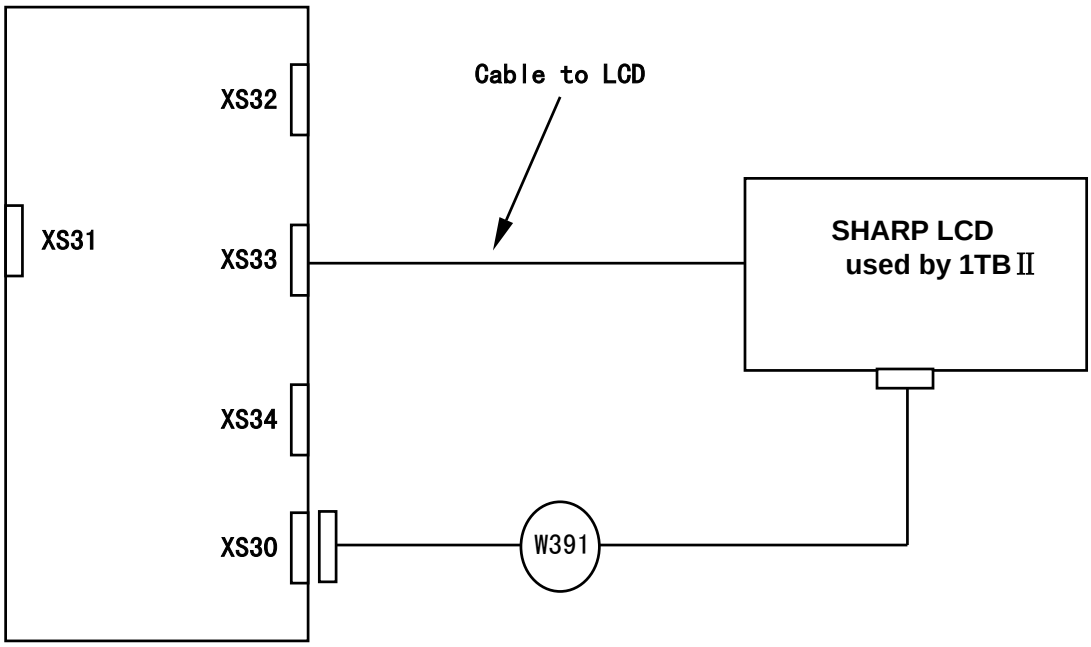


Fig. 2.2.2b

2.3 Setting switch description

2.3.1 Switch on main board

Name	Status		Explanation	Note
SA3	1，2	OFF	Default status	If 1,2 is set on,the system is reset.
	3，4	ON		
SA4		ON	Voltage of one rotation signal used for X returning reference position is +5V.	one rotation signal is fed by decoder
		OFF	Voltage of one rotation signal used for X returning reference position is +24V.	one rotation signal is fed by proximity switch
SA5		ON	Voltage of one rotation signal used for Z returning reference position is +5V.	one rotation signal is fed by decoder
		ON	Voltage of one rotation signal used for Z returning reference position is +24V.	one rotation signal is fed by proximity switch
SA6		1-2ON	Voltage of VP is +24V	Note 2
		2-3ON	Voltage of VP is +5V	
SA7		ON	Motion command output：Positive command pulse+negative command pulse	
		OFF	Motion command output：Command pulse+direction signal	
SA8		ON	K1TA CNC sysytem	
		OFF	K1TB CNC system	
SA9 SA10 SA11	SA9 SA10 SA11			SA9,SA10,SA11 should be set in the same status.
		ON	The signal from spindle decoder is sigle polarity	
		OFF	The signal from spindle decoder is double polarity	
T01~T04		ON	The DI is pulled up internal	Refer to 5-3
		OFF	The DI is not pulled up internal	
T05~T08 #DECX #DECZ		ON	The DI is pulled up internal	
		OFF	The DI is not pulled up internal	

Note：**1. <u>The status with blck backgroud is default</u>，the user can change it to meet the actual requirement .**

2. VP indicates the voltage fed to the driver. The VP is output at pin 12,13 of socket XS51(X axis) and XS52(Z axis).

3. If KND-BD3A，KND-BD3Y，KND-BD3H step motor driver are adopted, VP=+5V.
If KND-BD5L is adopted, no VP is required.
If KND-SD98 AC servo driver is adopted, VP=+24V.

4. Motion command format of KND series driver：Command pulse+direction signaKND.

2.4 Connection between main board and back cover

2.4.1 Connection between XS5 and XS50,XS51

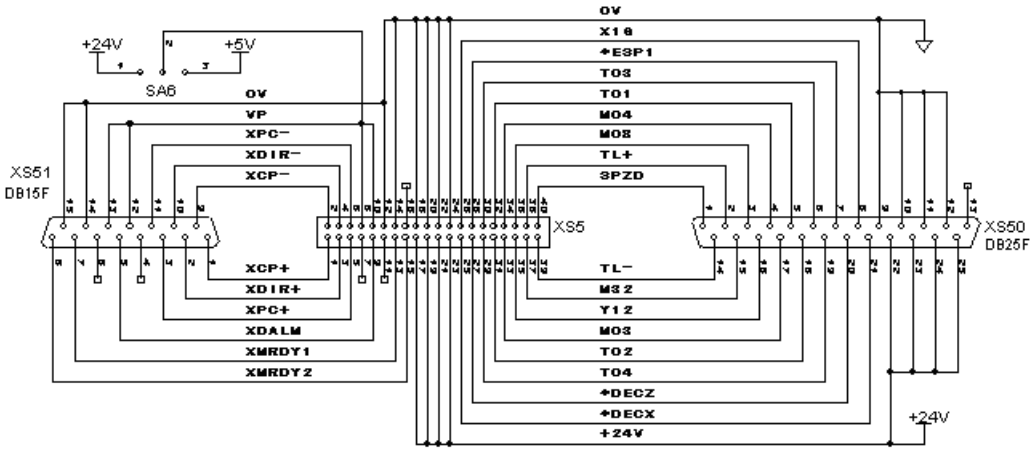


Fig. 2.4.1

2.4.2 Connection between XS6 and XS52,XS53

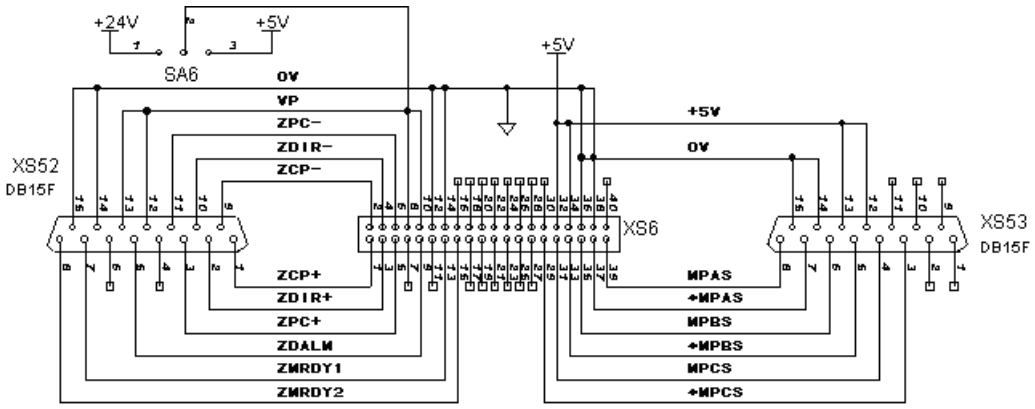


Fig. 2.4.2

2.4.3 Connection between XS7 and XS54，XS55,XS56

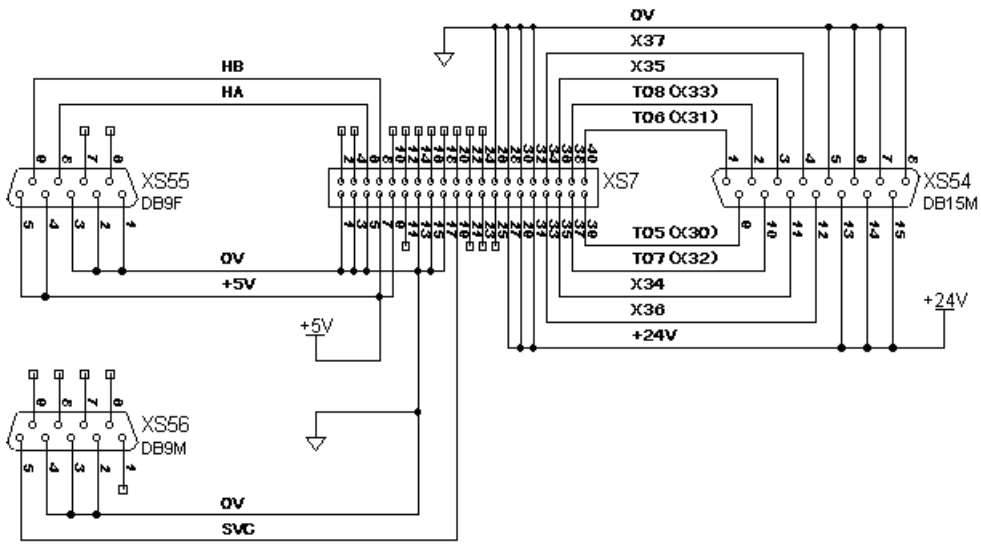


Fig. 2.4.3

2.4.4 Connection between XS10 and XS57

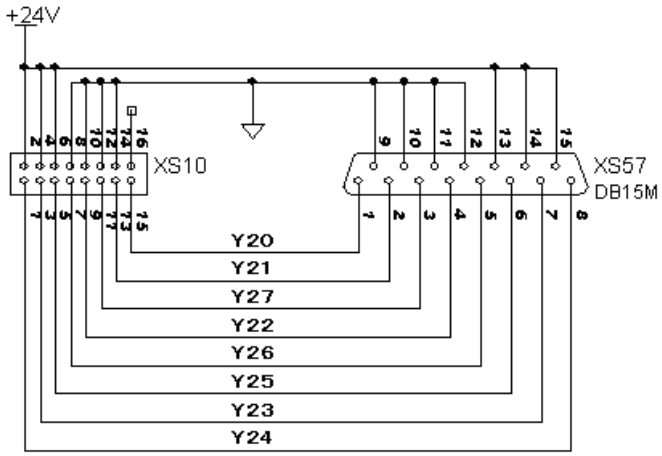


Fig. 2.4.4

2.4.5 Connection between XS1 and the power unit

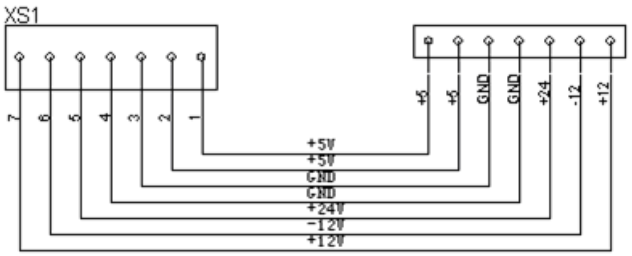
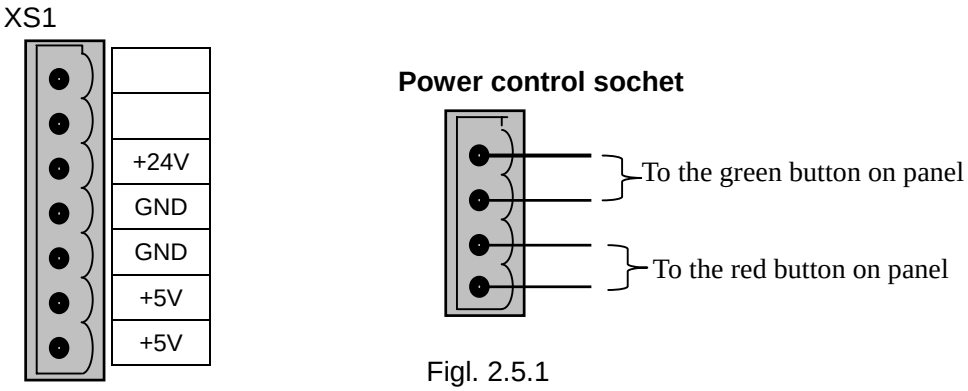


Fig. 2.4.5

2.5 Socket on the back cover

2.5.1 Power socket XS1 and Power control sochet



Figl. 2.5.1

2.5.2 Back cover Diagram

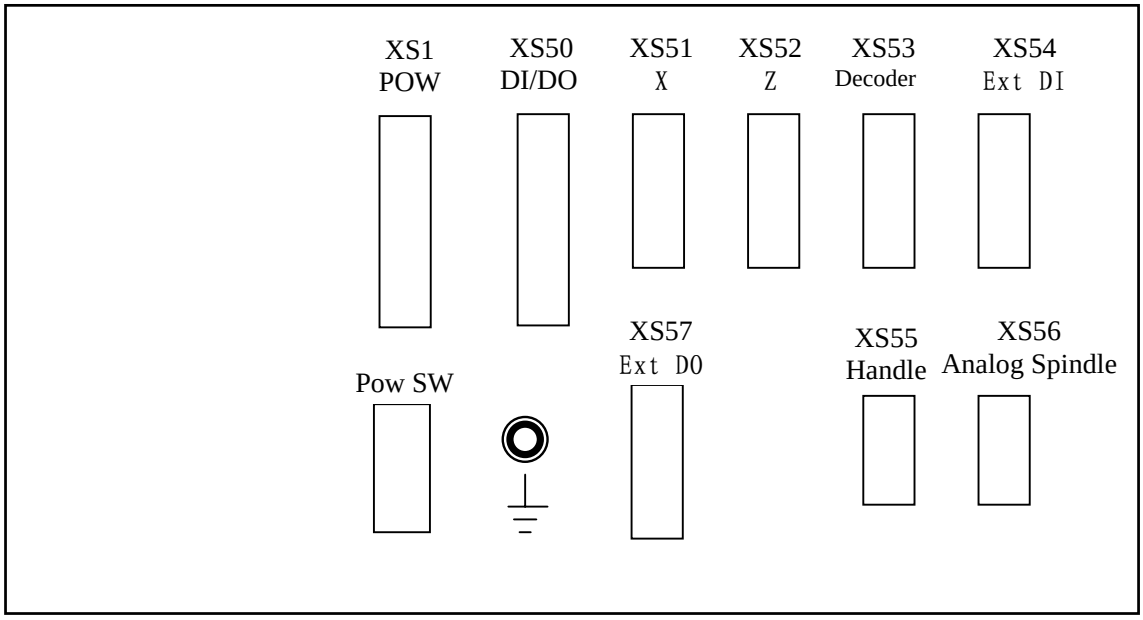


图 2.5.2

Note：Please connect the ground line to the marked position.

3 EXTERNAL CONNECTION

3.1 System External Connection Diagram

3.1.1 Connection Diagram (stepper motor)

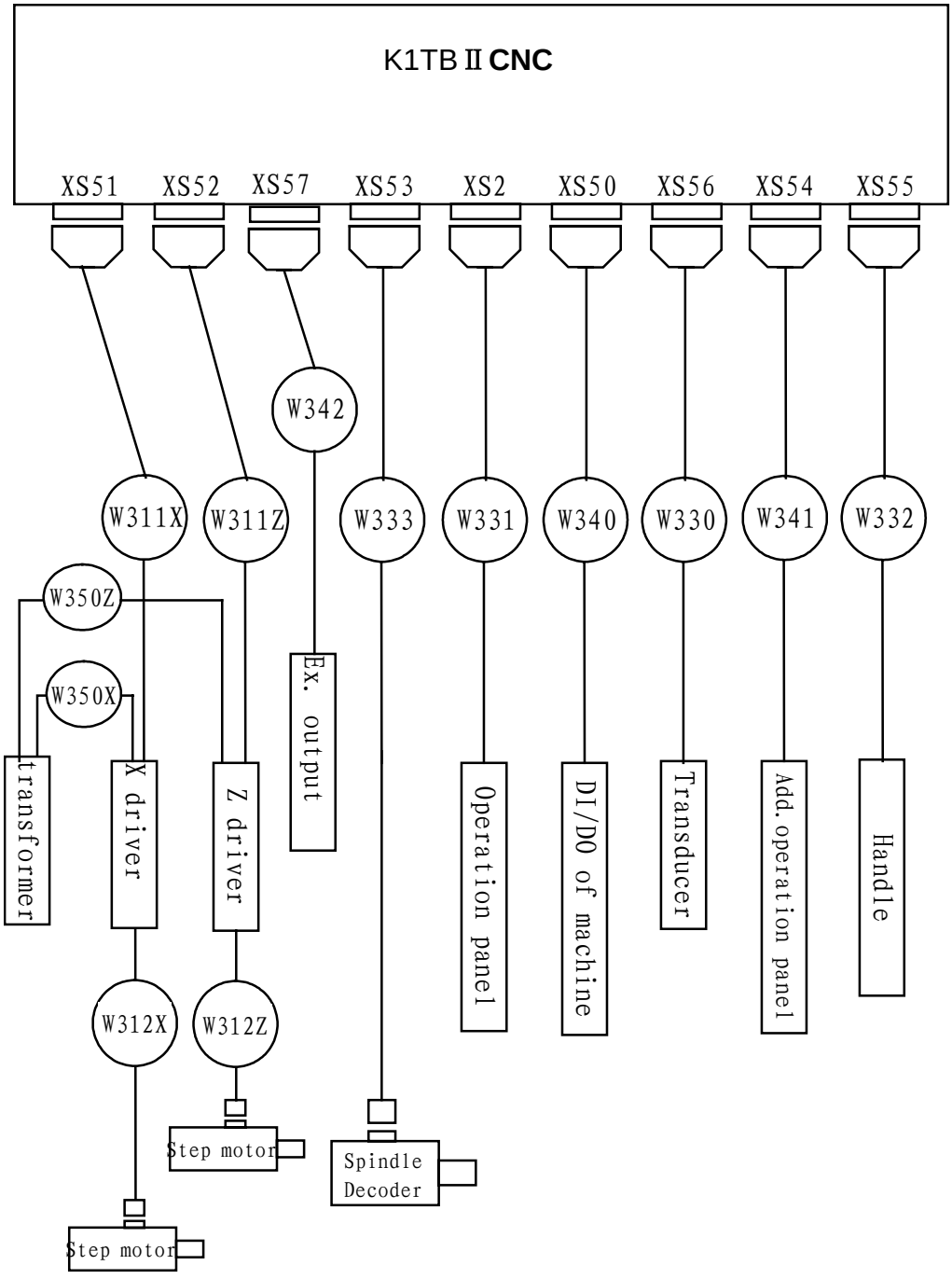
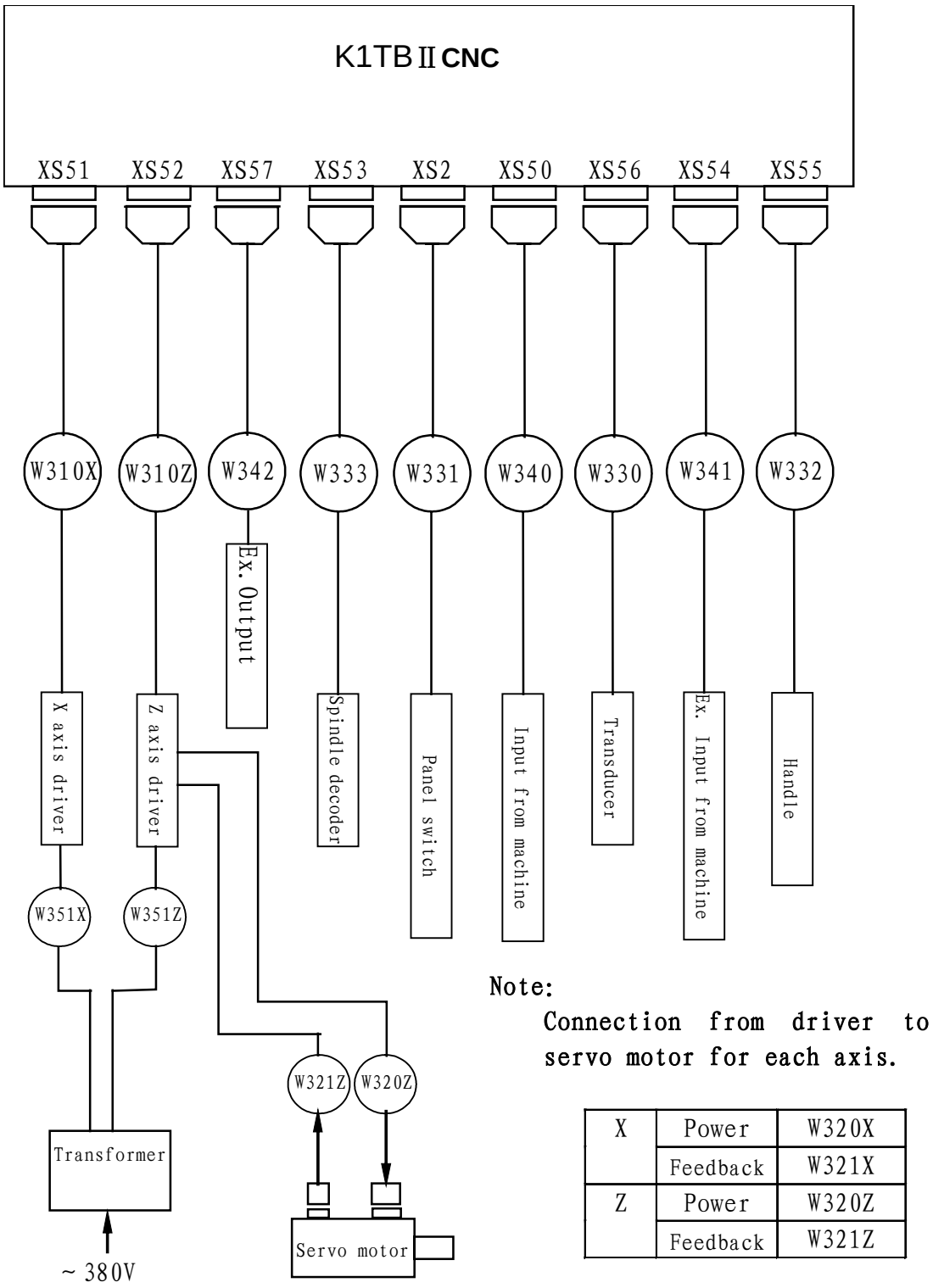


Fig. 3.1

3.1.2 Connection Diagram (digit AC servo motor)



Note:

Connection from driver to servo motor for each axis.

X	Power	W320X
	Feedback	W321X
Z	Power	W320Z
	Feedback	W321Z

Fig. 3.2

3.2 Interface Signal from CNC to Driver

3.2.1 Interface Signal Diagram

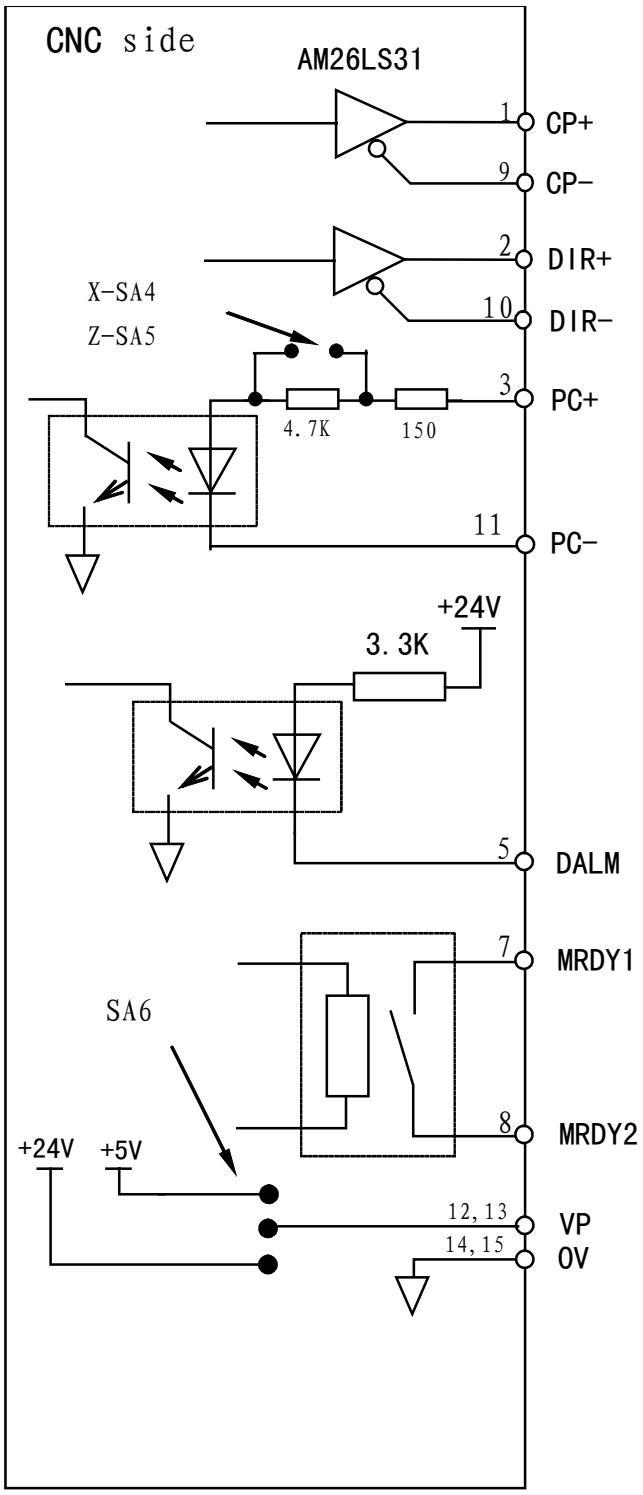


Fig. 3.2.1

3.2.2 Connector Signal List

XS51:DB15F（X axis）

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		

XS52:DB15F（Z axis）

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		

Fig. 3.2.2

3.2.3 Signal Description (n:X/Y)

(1) motion command signal

(a) Single pulse output (SA7 break)

nCP＋，nCP－；nDIR＋，nDIR－。

NCP: command pulse signal，nDIR: motion direction signal. They are all differential output.

(b) Double pulse output (SA7 short)

nCP: CCW command pulse signal，nDIR: CW command pulse signal

(c) The circuit of motion command signal

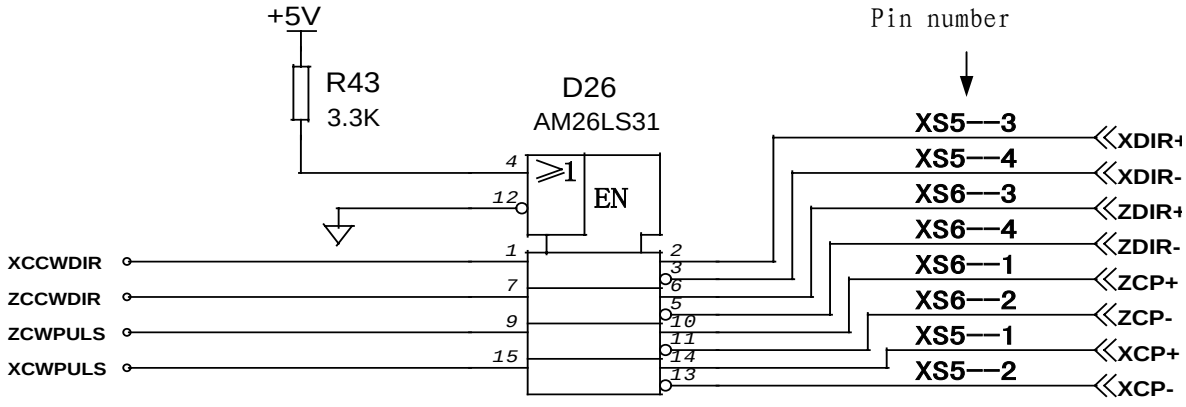


Fig. 3.2.3a

（2）Zero point signal of machine tool reference position

Receive circuit：

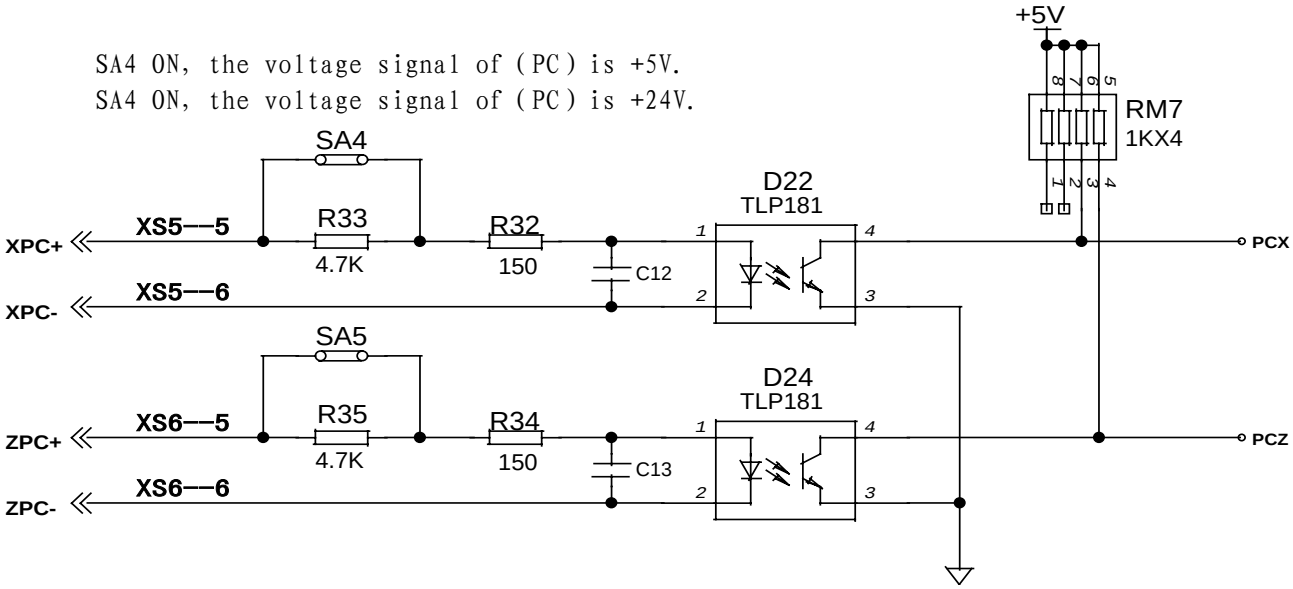


Fig. 3.2.3b

nPC or nPC+ signal wave provided by user is as below:

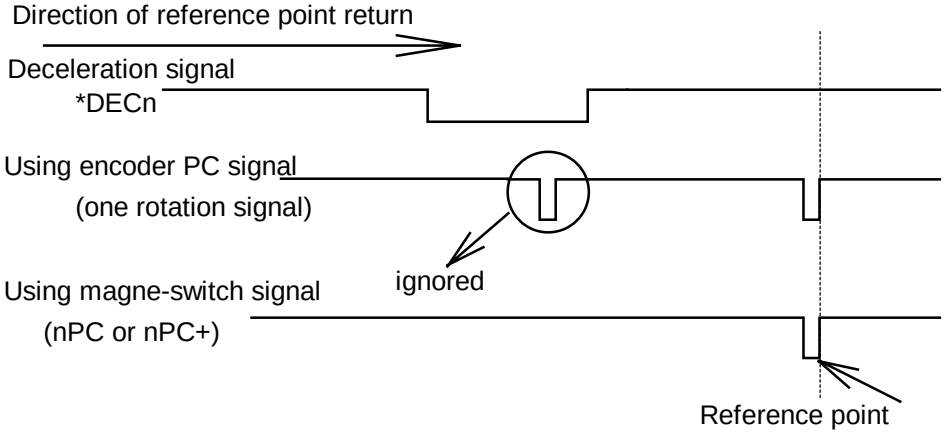


Fig. 3.2.3c

The connection diagram using a magne-switch as deceleration switch of zero return is as below:

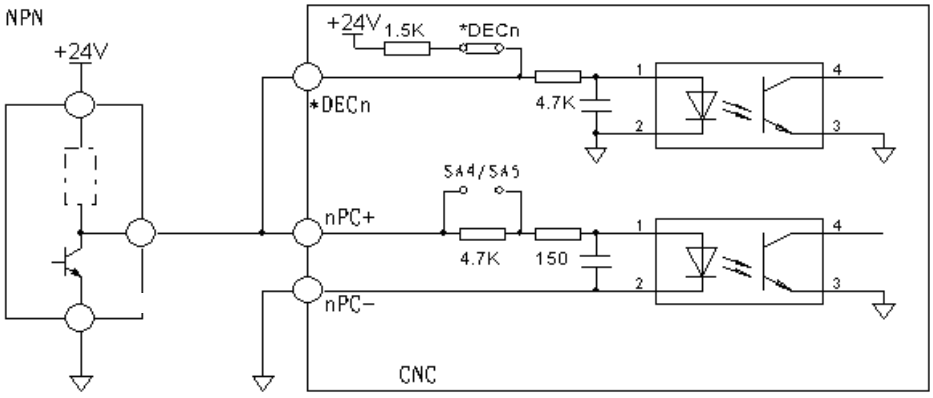


Fig. 3.2.3d

The connection diagram using a magne-switch as deceleration switch and zero signal of zero return is as below:：

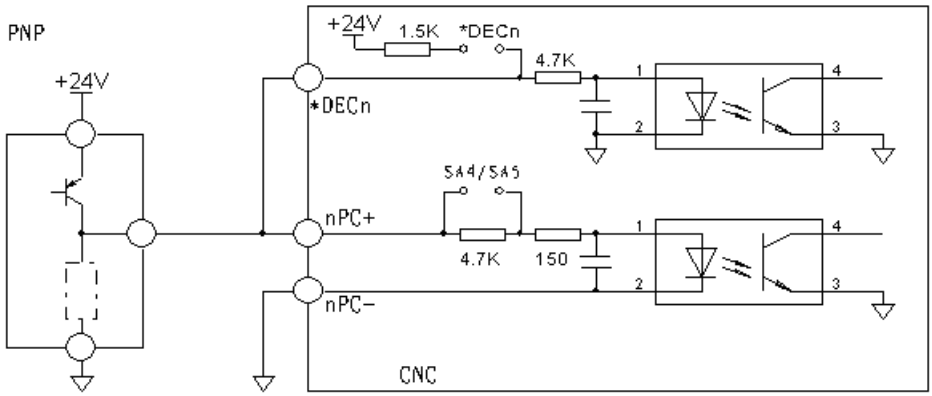


Fig. 3.2.3e

The reference position return is described if Fig. below：

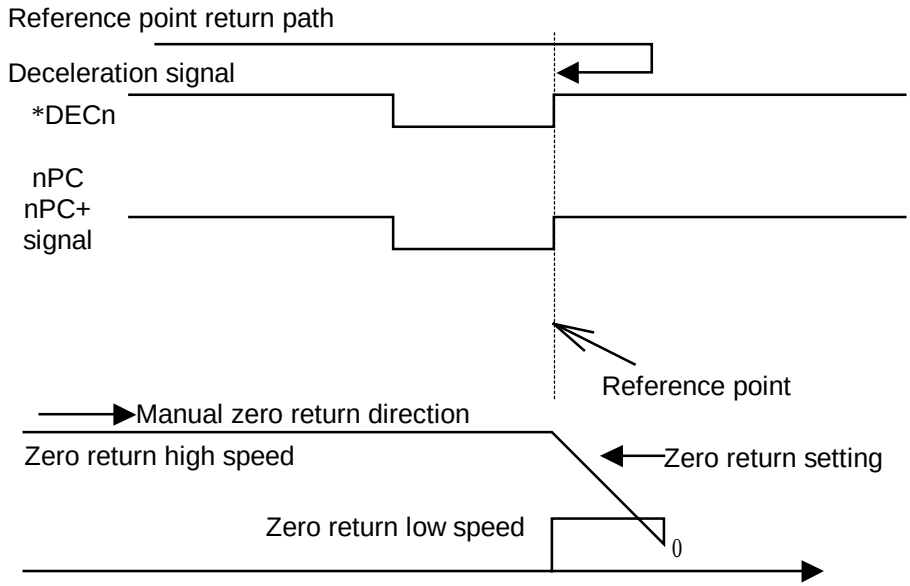


Fig. 3.2.3f

Note：This is zero return mode C,the parameter ZRSX/Z，ZCX/Z should be set “1”

（3）Driver device alarm signal nDALM (input)

The effective voltage level of input signal can be set to high voltage level or low voltage level by parameter DALX/DALZ. This kind of input circuit requires that driver device can provide the following signal:

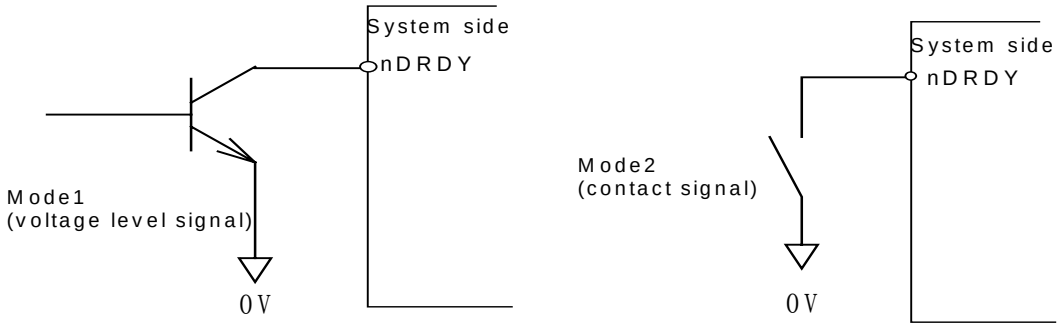


Fig. 3.2.3g

receive circuit：

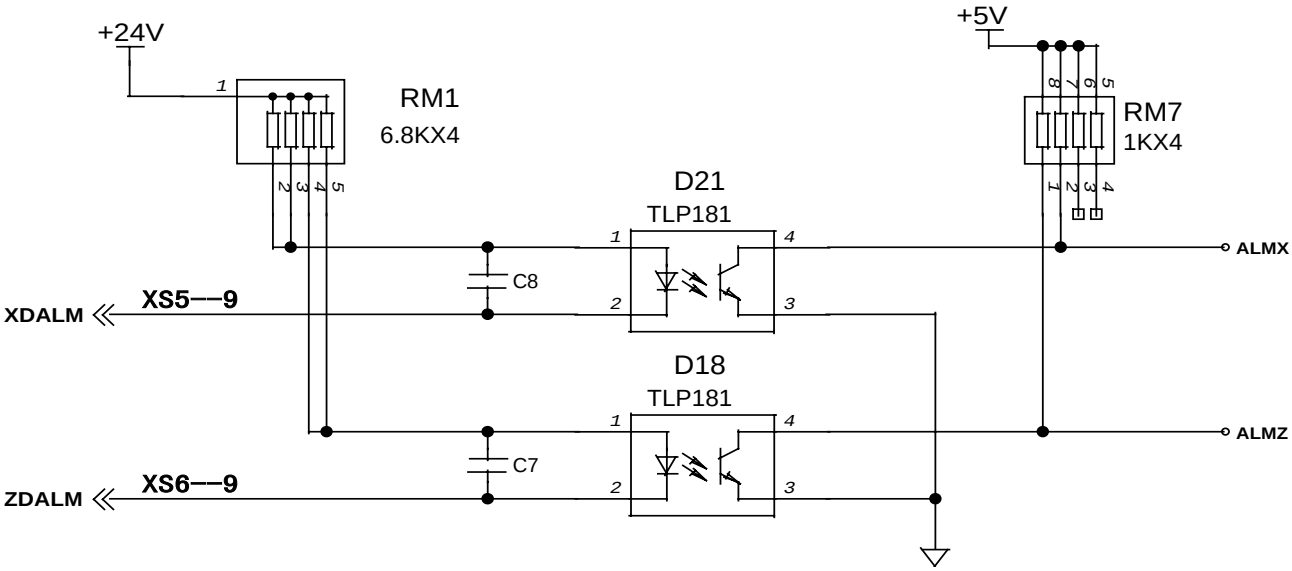


Fig. 3.2.3h

（4）)CNC system ready signal nMRDY1、nMRDY2（relay contact output）

Output circuit

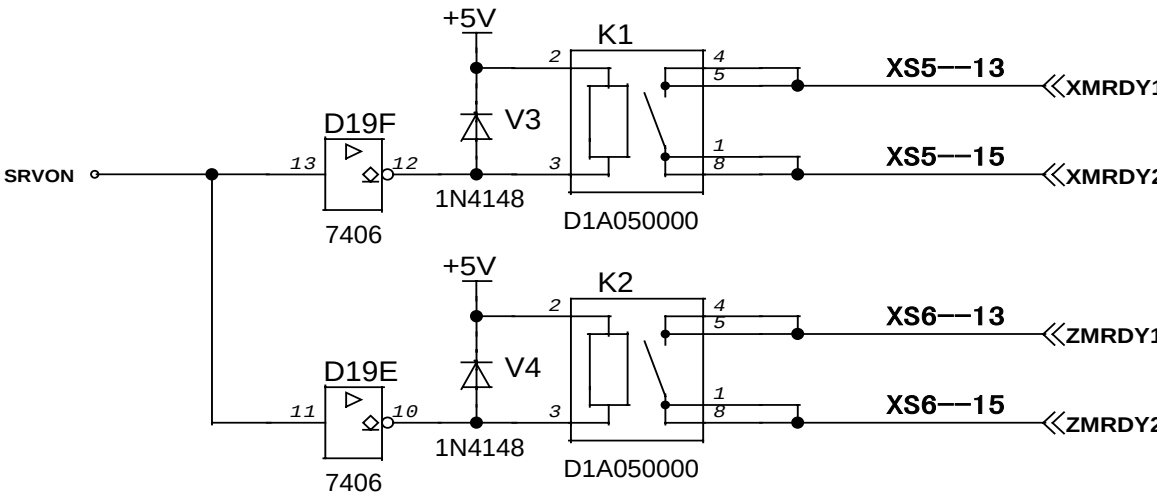


Fig. 3.2.3i

This contact will close after CNC initialization normal. If driver device alarm or emergency is examined by CNC, this contact will break.

（5）VP is a kind of driver device voltage type provided by CNC system（+5V or +24V）. It is determined by the setting of the first, the second, or the third bit of SA6 setting switch.

3.2.4 Description of Cable

（1）Cable making of connecting KND—BD5L series stepper motor driver

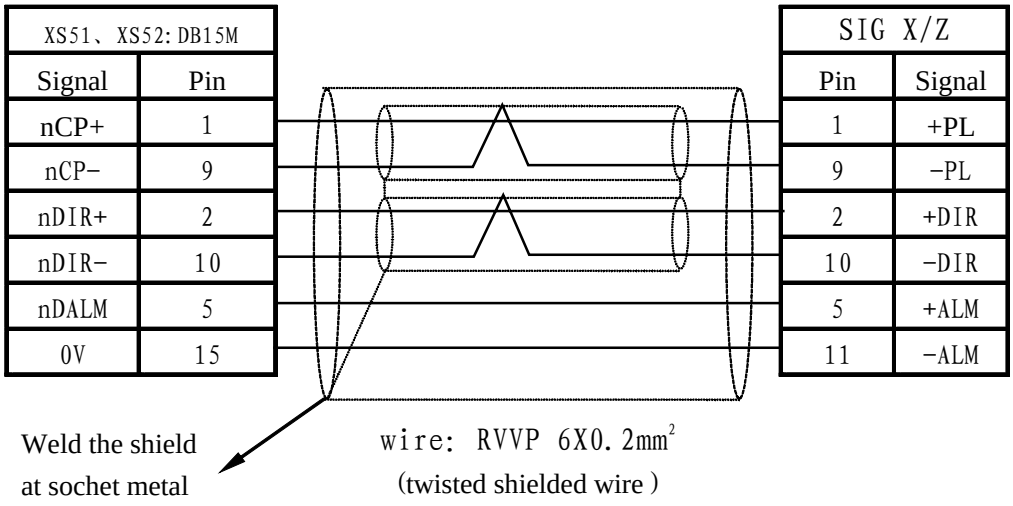


Fig. 3.2.4a

（2）Cable making of connecting KND—BD3H series stepper motor driver

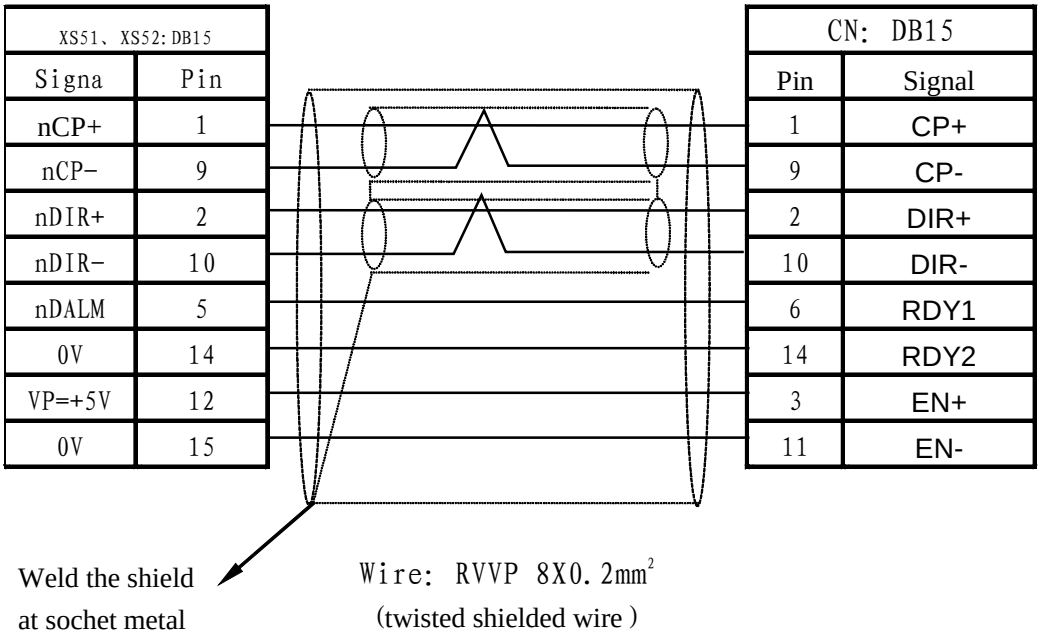


图 3.2.4b

（3）Cable making of connecting KND—BD3A series stepper motor driver

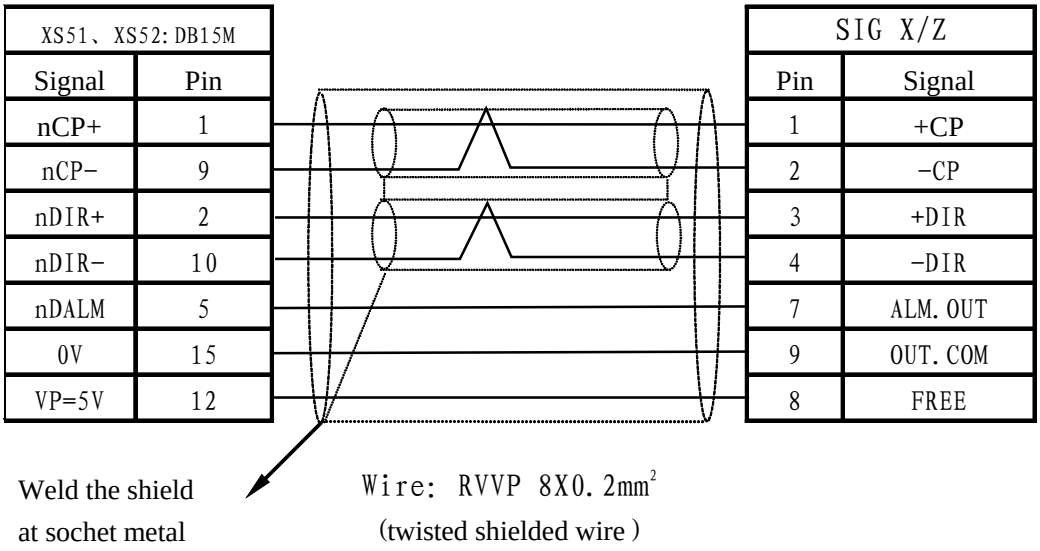


Fig. 3.2.4c

（4）Cable making of connecting KND—BD3Y series stepper motor driver

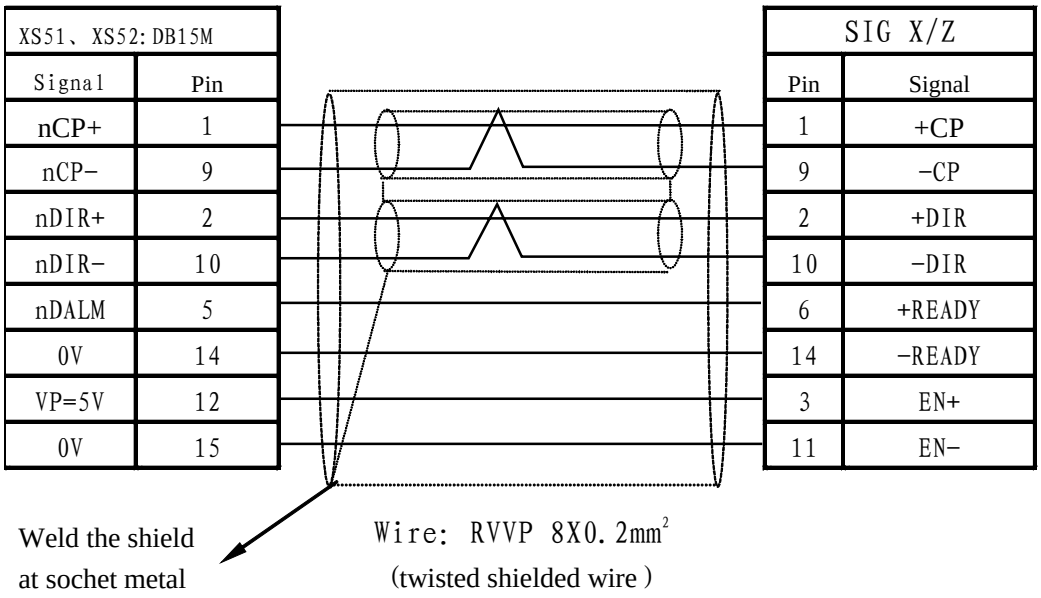


Fig. 3.2.4d

（5）Cable making of connecting JAPAN YASKAMA AC servo driver

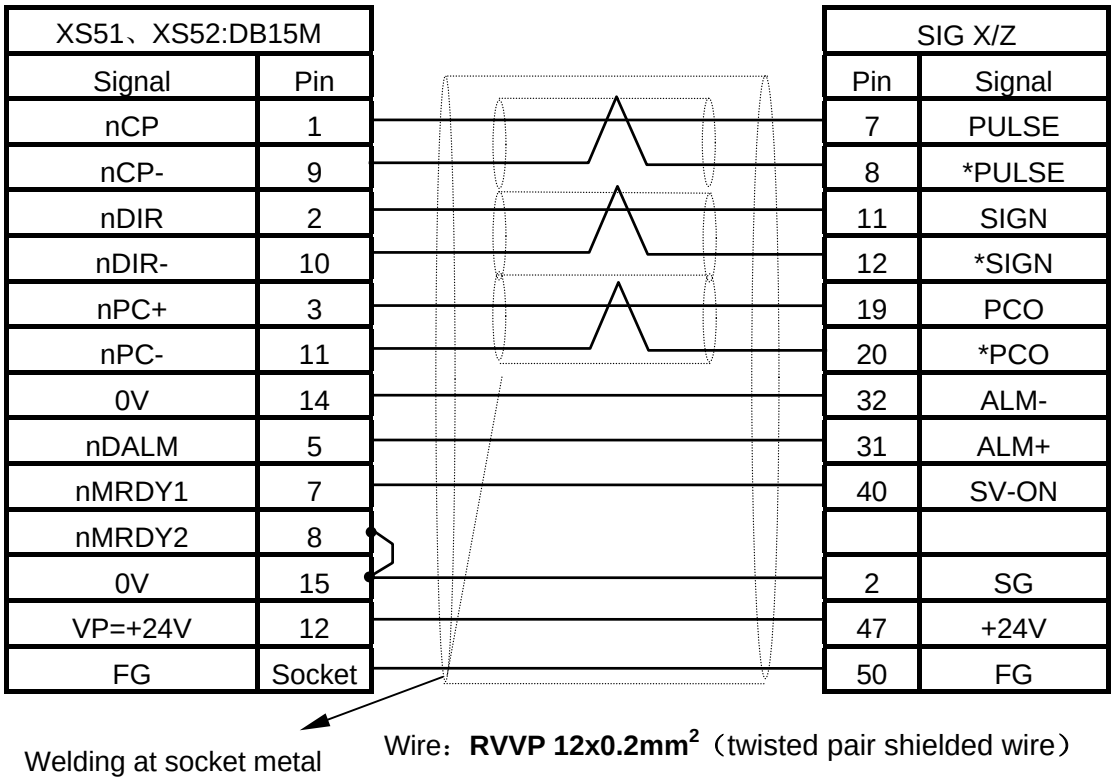


Fig. 3.2.4e

（6）Cable making of connecting JAPAN Panasonic MINAS - A AC servo driver

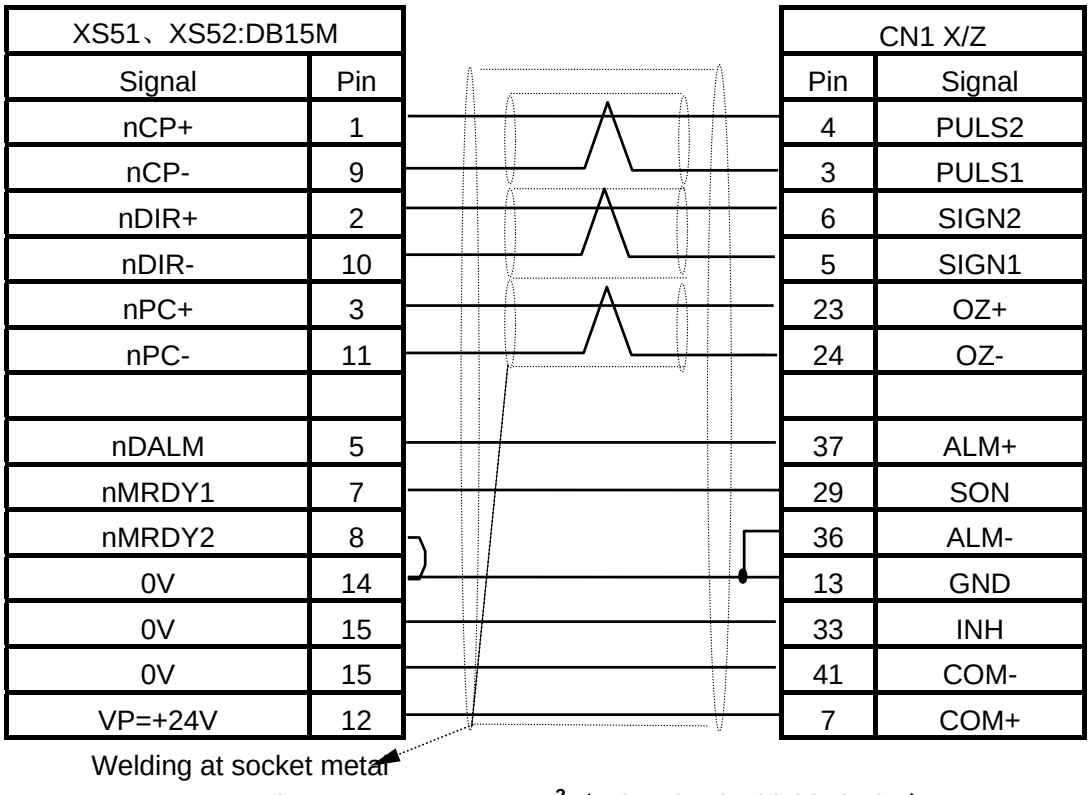
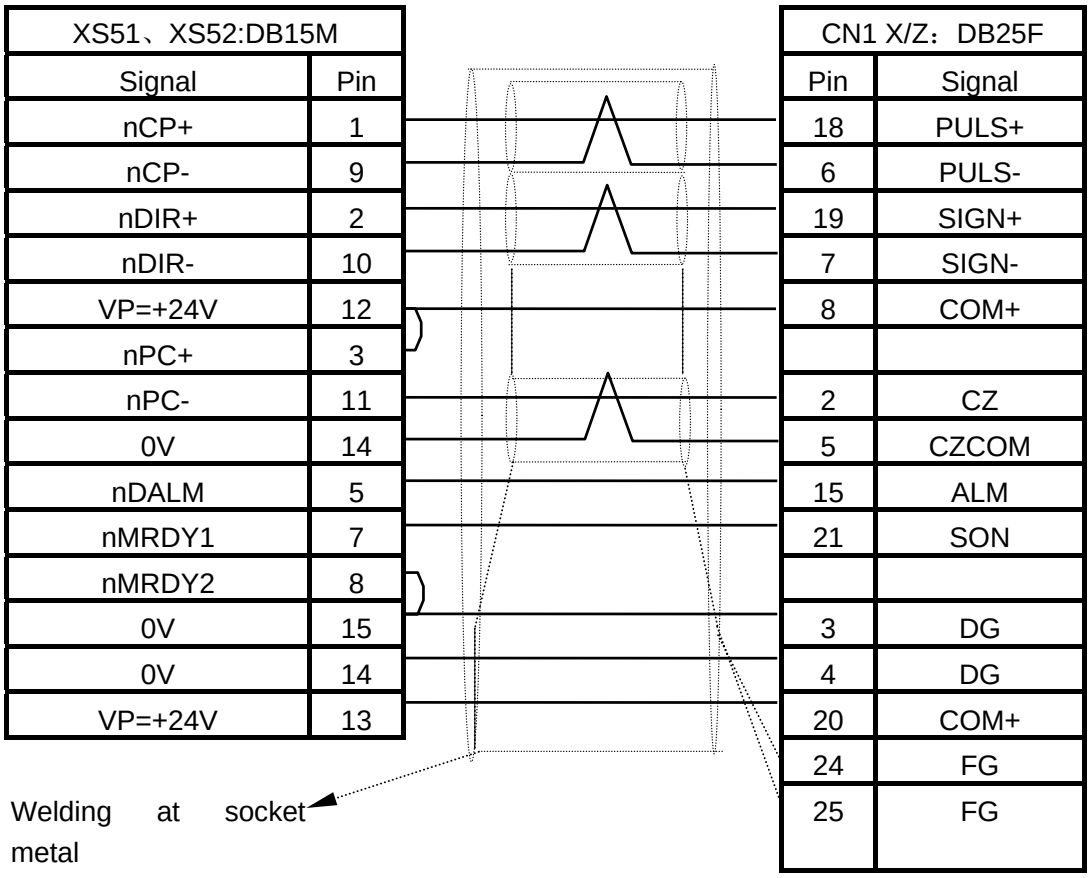


Fig. 3.2.4f

（7）Cable making of connecting KND—SD98 AC servo driver



Wire：**RVVP 12x0.2mm^2**（twisted pair shielded wire）

Fig. 3.2.4g

3.3 Connection of Spindle position decoder

3.3.1 Signal list of Spindle position decoder

XS53: DB15F

1	NC	9	NC
2	NC	10	NC
3	*MPCS	11	NC
4	MPCS	12	+5V
5	*MPBS	13	+5V
6	MPBS	14	0V
7	*MPAS	15	0V
8	MPAS		

Fig. 3.3.1

3.3.2 Receive circuit：

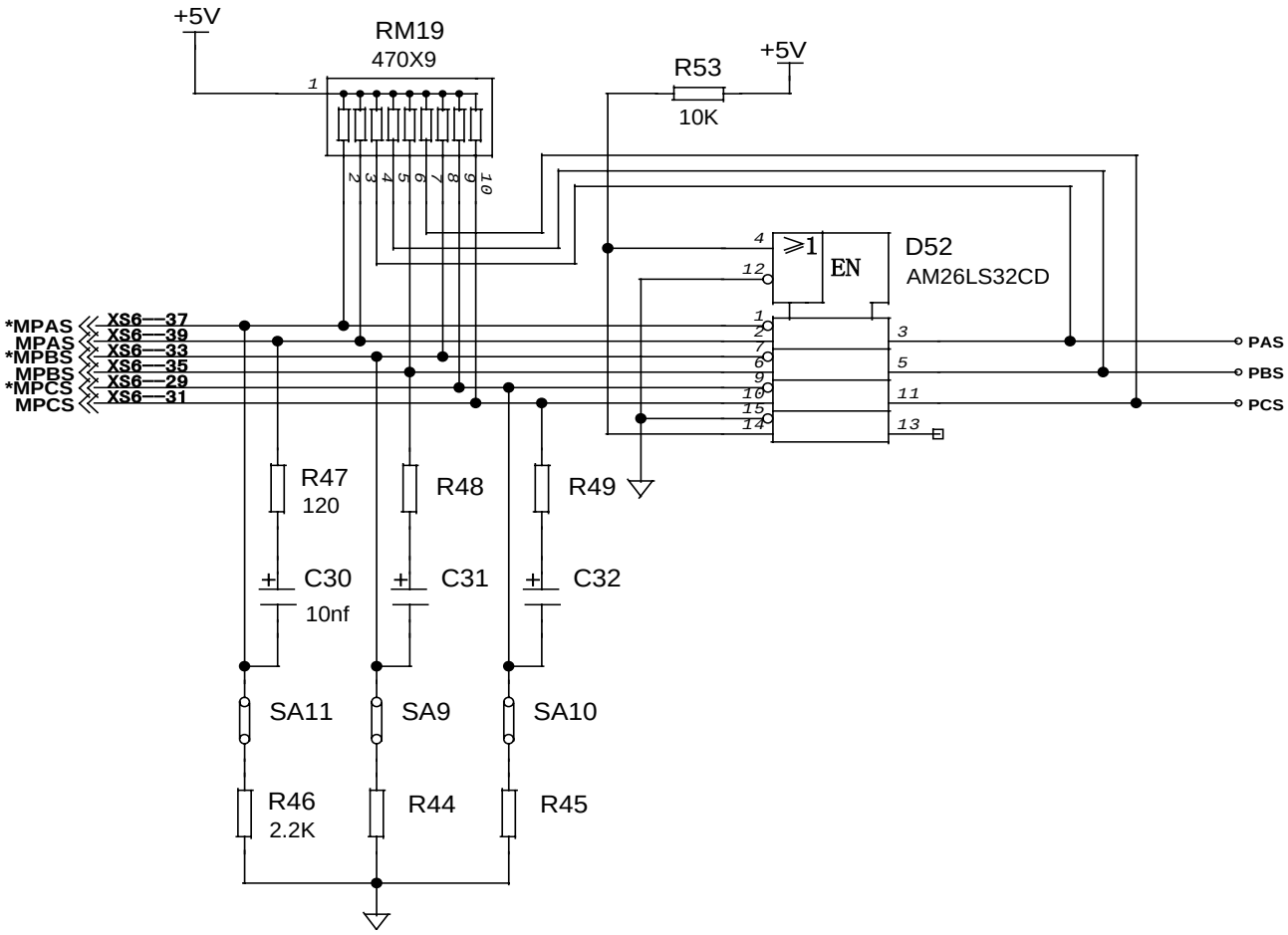
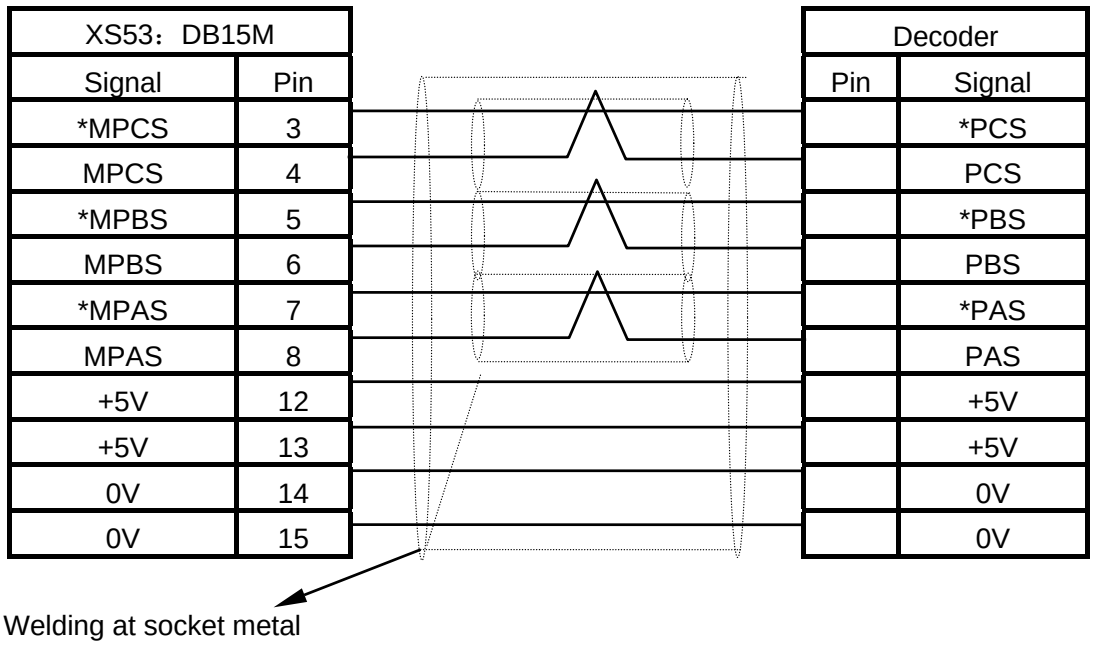


Fig. 3.3.2

The circuit is default from factory。The default Spindle position decoder is ：LF-102.4BM-C05D，1024 pulse/rotation，Voltage +5V

3.3.3 Cable making of connecting spindle position decoder



Wire：**RVVP 10x0.2mm^2**（twisted pair shielded wire）

Fig. 3.3.3

3.4 Connection of Handle

3.4.1Receive circuit：

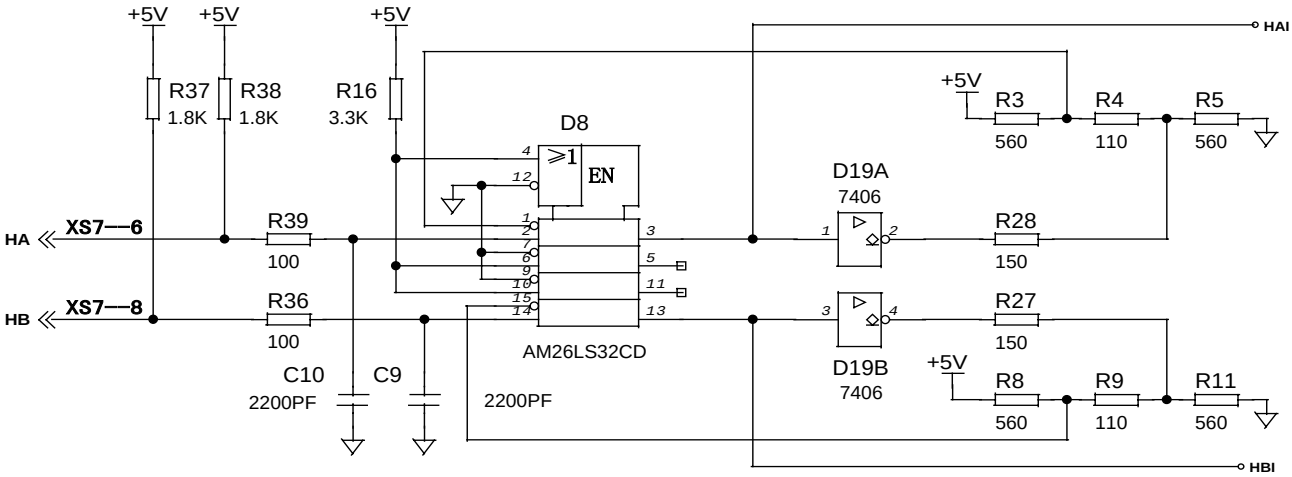


Fig. 3.4.1

3.4.2 Signal list of Handler

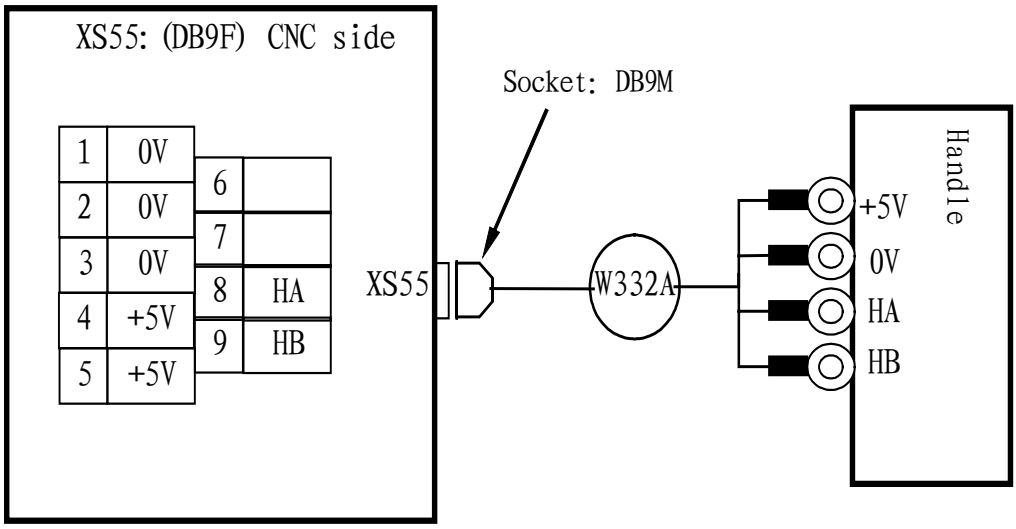


Fig. 3.4.2

The default Handler is LGF-001-100，100 pulse/rotation，Voltage +5V

3.5 Connection of analog spindle

3.5.1output circuit

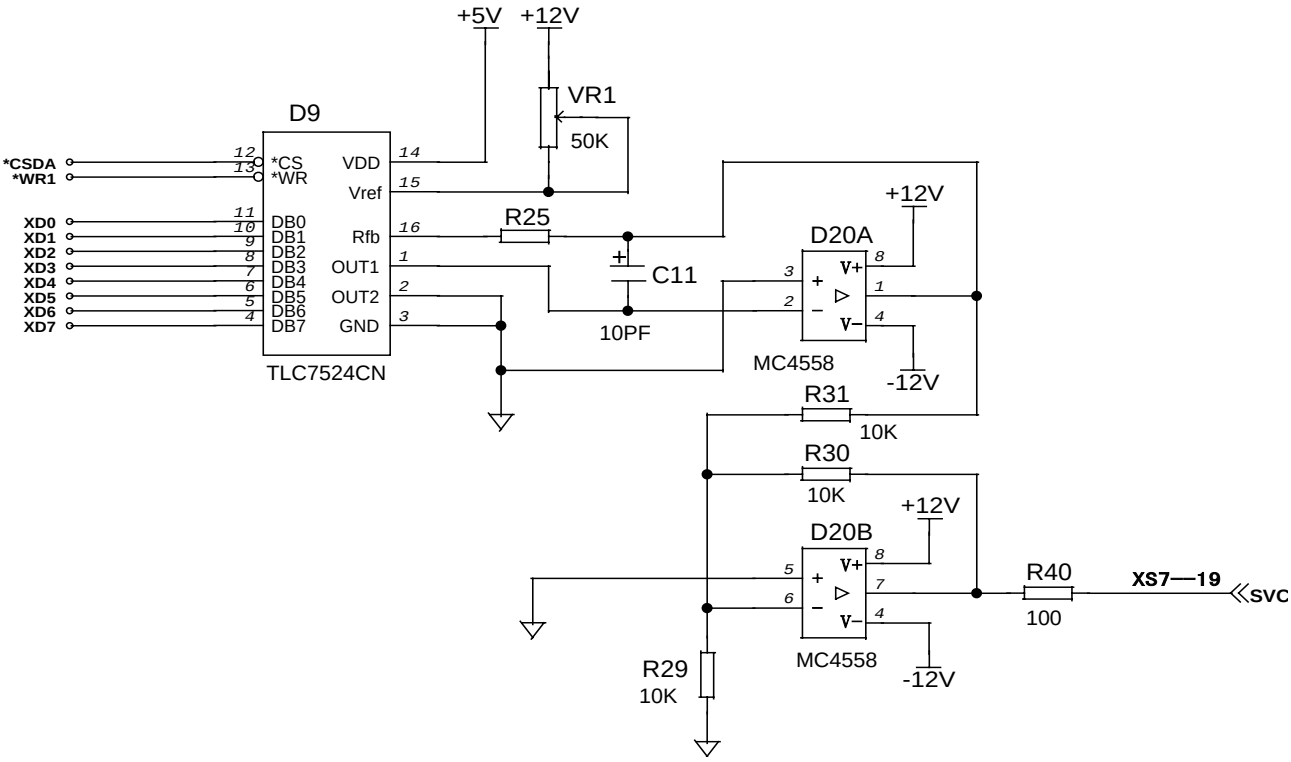


Fig. 3.5.1

3.5.2 Signal list of analog spindle

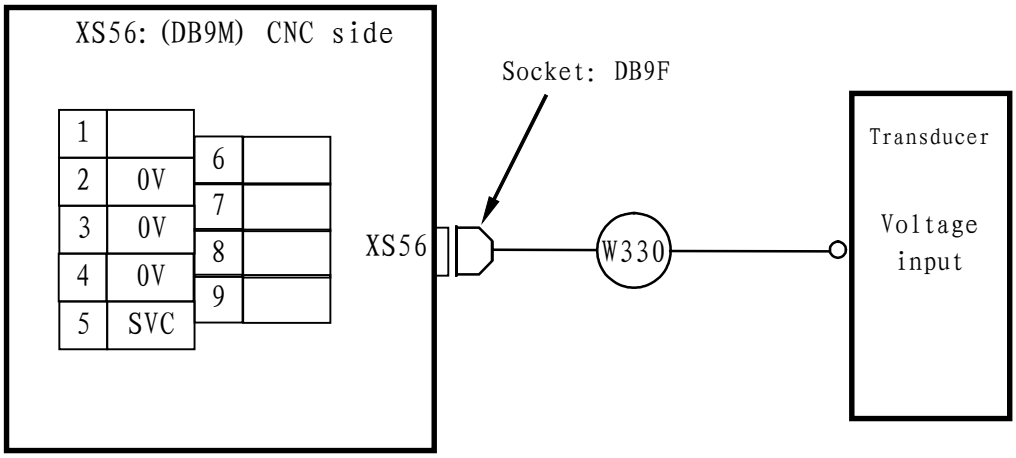


Fig. 3.5.2

Note：recommended wire- RVVP2X0.5mm^2（twisted pair shielded wire）

3.6 Connection of Operator's Panel and Additional Operator's Panel

3.6.1 Signal List of Connector

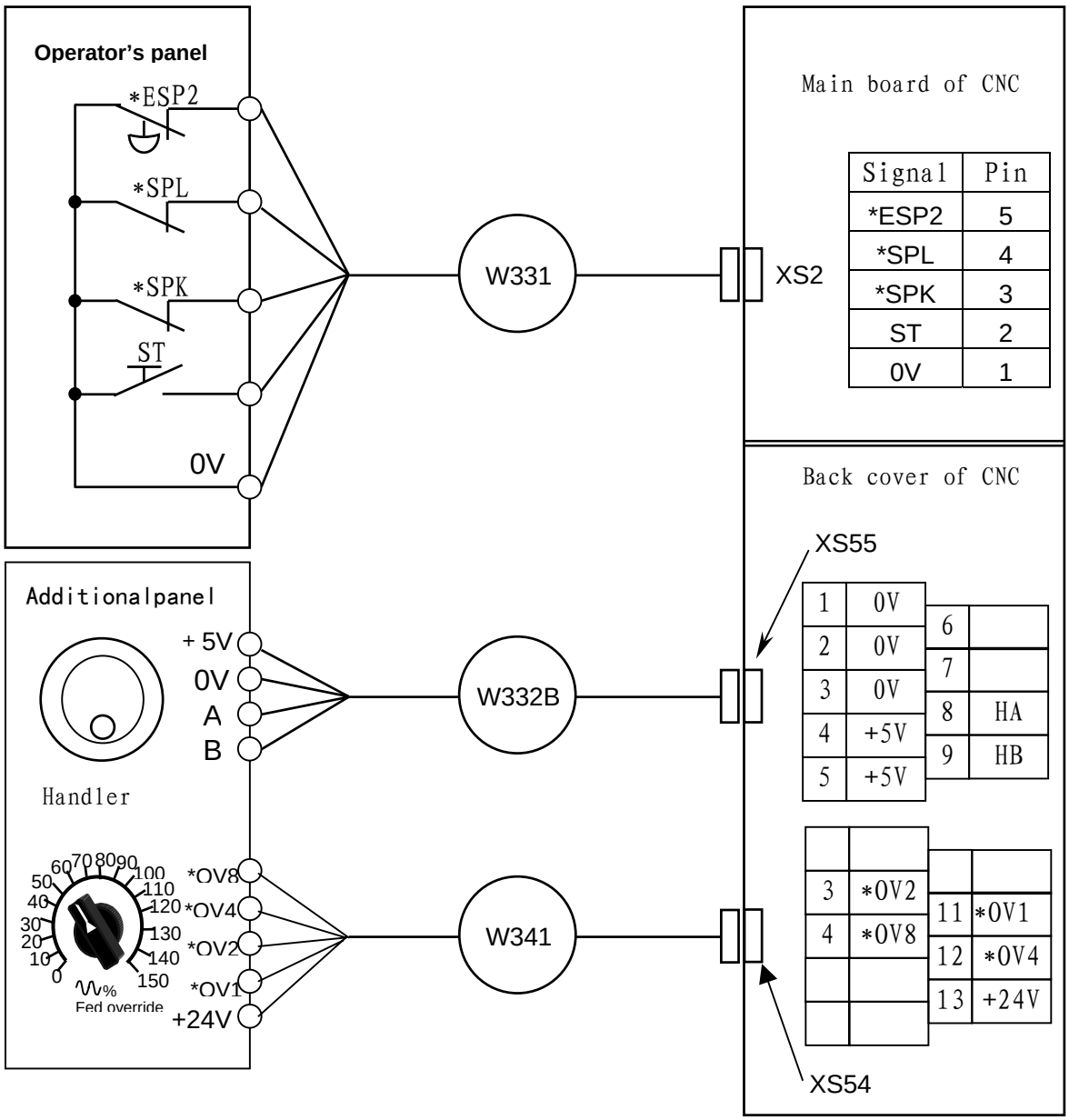


Fig. 6.1

3.6.2 Signal Description

（1） *ESP2：Emergency stop , is the same as *ESP1.
（2） ST： Cycle start
（3） *SPK： Feedhold
（4） *SPL： Spindle hold
（5） 0V： Signal common

3.6.3Receive circuit

DI signal circuit:

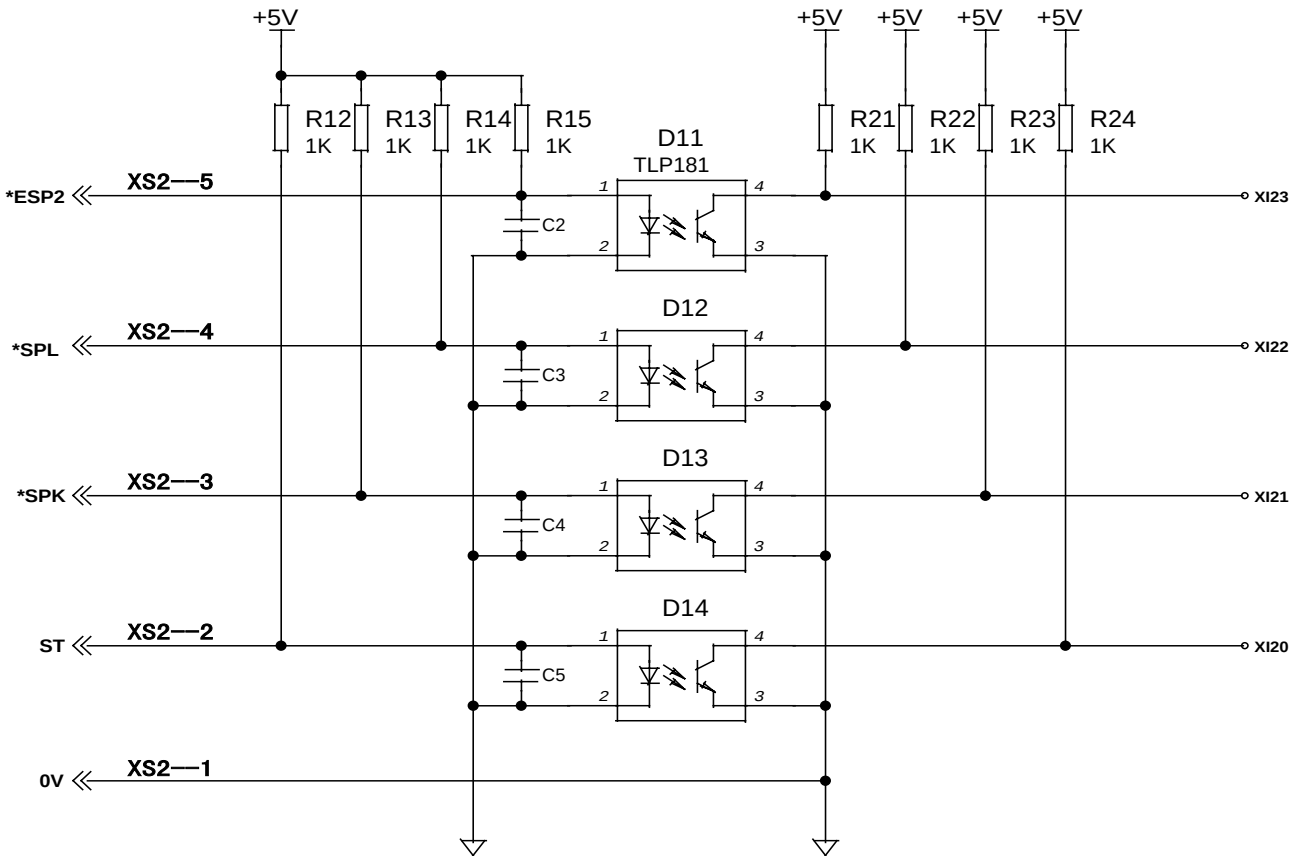


Fig. 6.3

3.7 Connection of transformer

3.7.1Connection between the transformer BK-1.3 and step motor driver

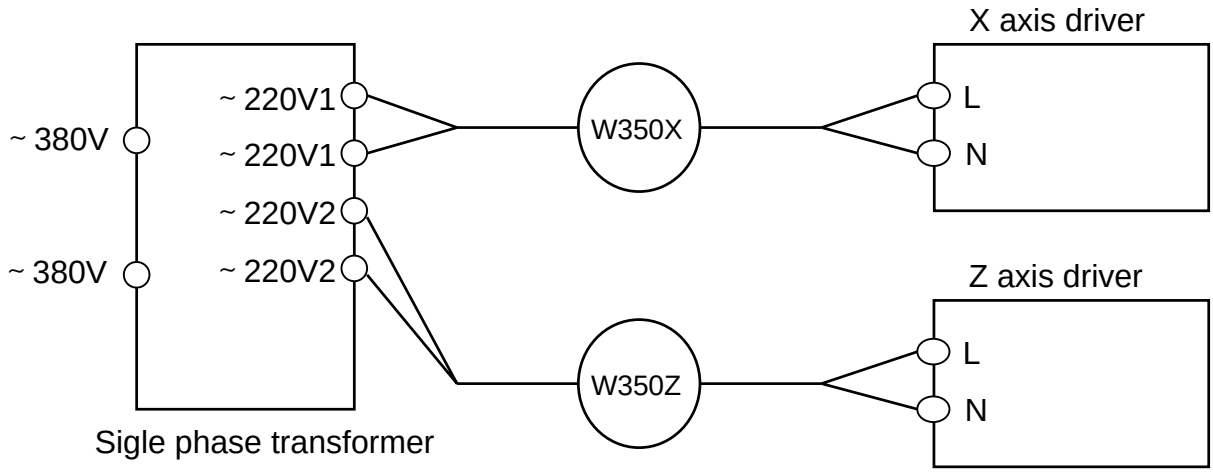


Fig. 3.7.1

3.7.2 Connection between the transformer SSG-3/0.5 and digital AC servo driver

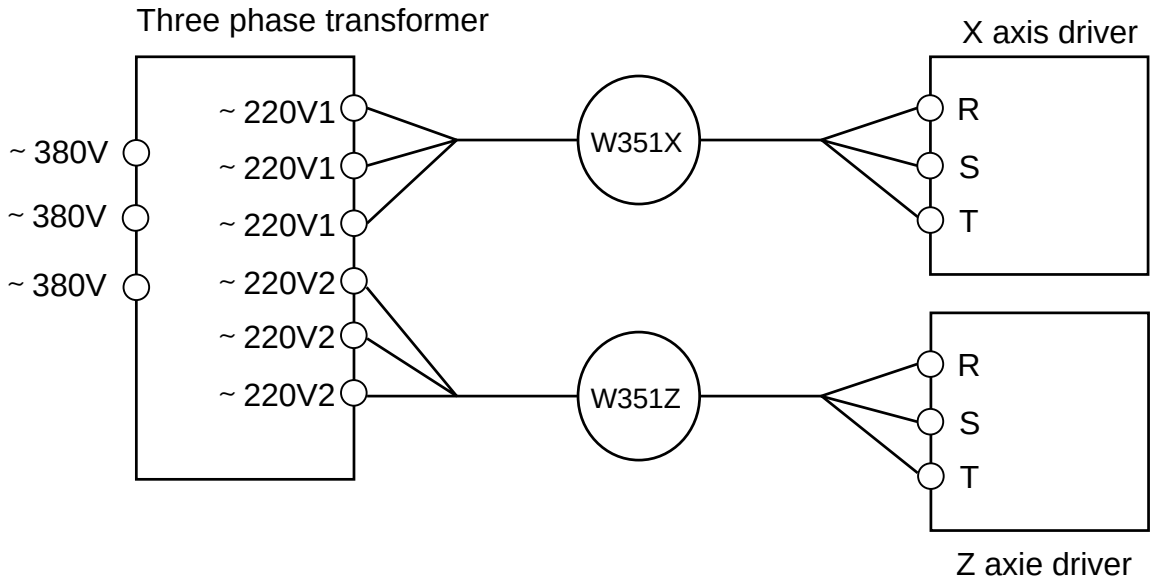


Fig. 3.7.2

4 MACHINE TOOL INTERFACE

4.1 DESCRIPTION OF INPUT SIGNAL INTERFACE

4.1.1 DC Input Signal A

DC input signal A is a signal from machine tool to CNC. They include push-key, limitation switch, relay contact and approach switch which come from machine tool side.

(1) Contact on machine tool according to the following conditions：

a. contact capacity ： more than DC30V、16MA

b. Leakage current in open circuit is less than 1MA（Vmax＝26V）

c. Contact voltage-drop in closed circuit is less than 2V（current 8.5MA, including cable voltage-drop）.

Signal circuit diagram is as Fig. 4.1.1.

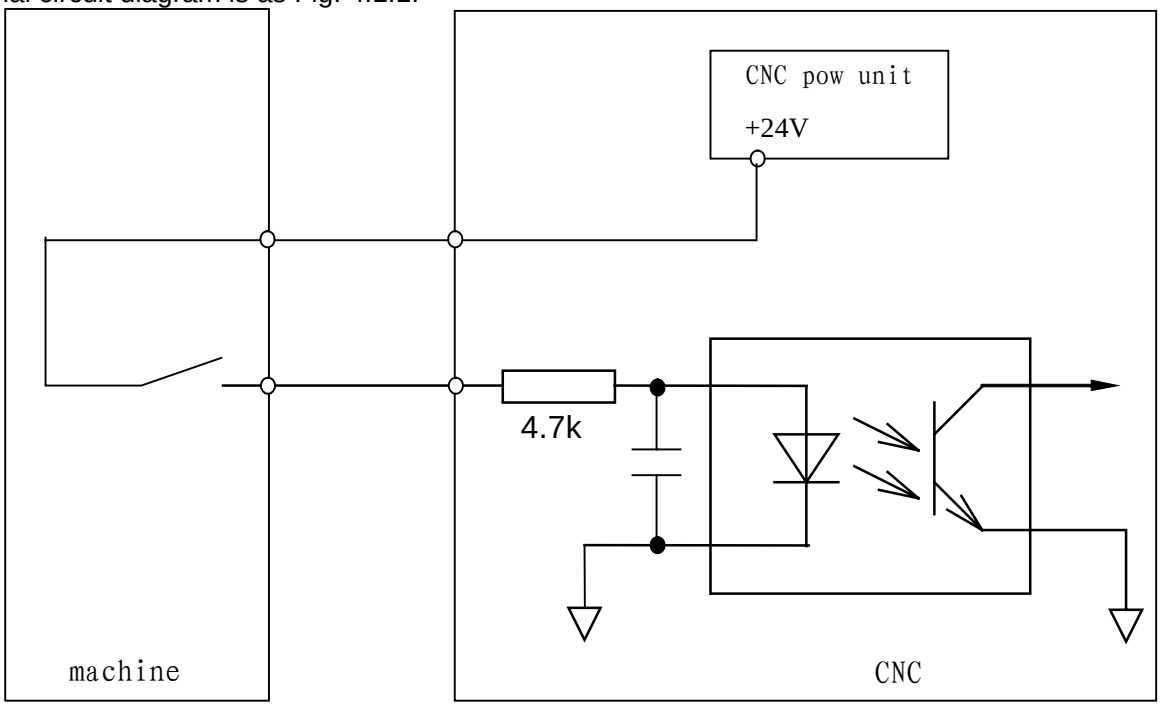


Fig. 4.1.1

4.1.2 DC Input Signal B

DC input signal B is a signal from machine tool to CNC. It is used at high speed.

(1) Contact on machine tool according to the following conditions：

a. contact capacity ： more than DC30V、16MA

b. Leakage current in open circuit is less than 1MA（Vmax＝26V）

c. Contact voltage-drop in closed circuit is less than 2V（current 8.5MA, including cable voltage-drop）.

(2) Signal circuit diagram is as Fig. 4.1.2a. or Fig. 4.1.2b.

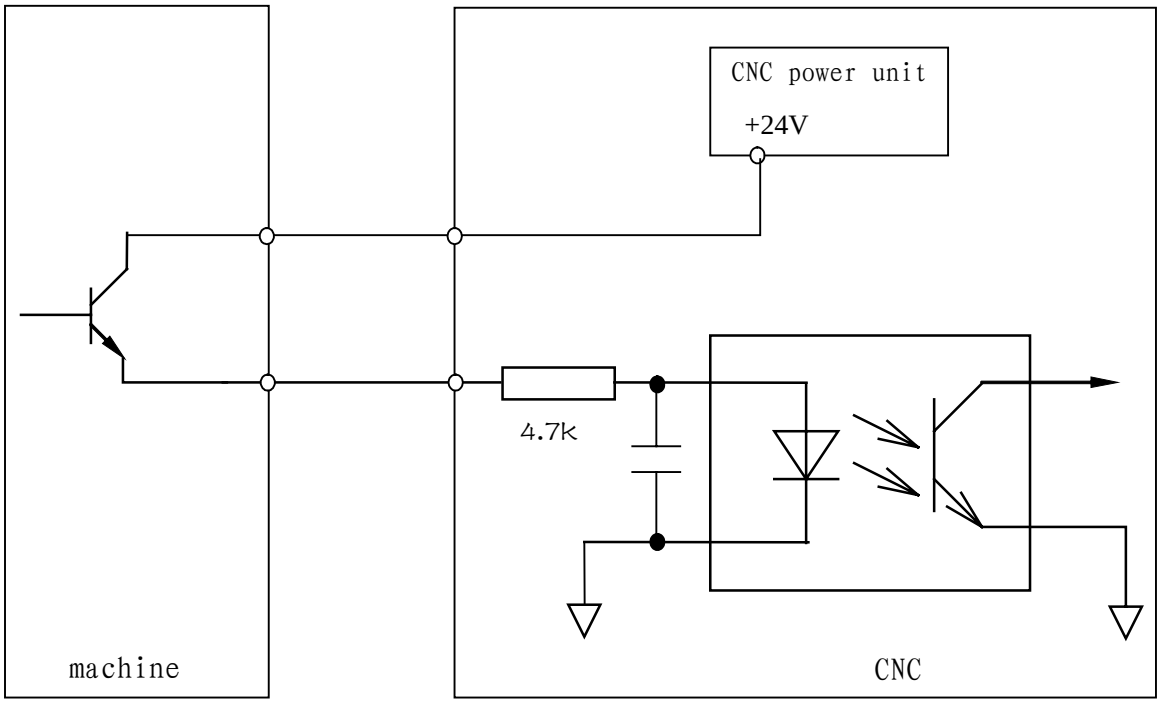


Fig. 4.1.2a

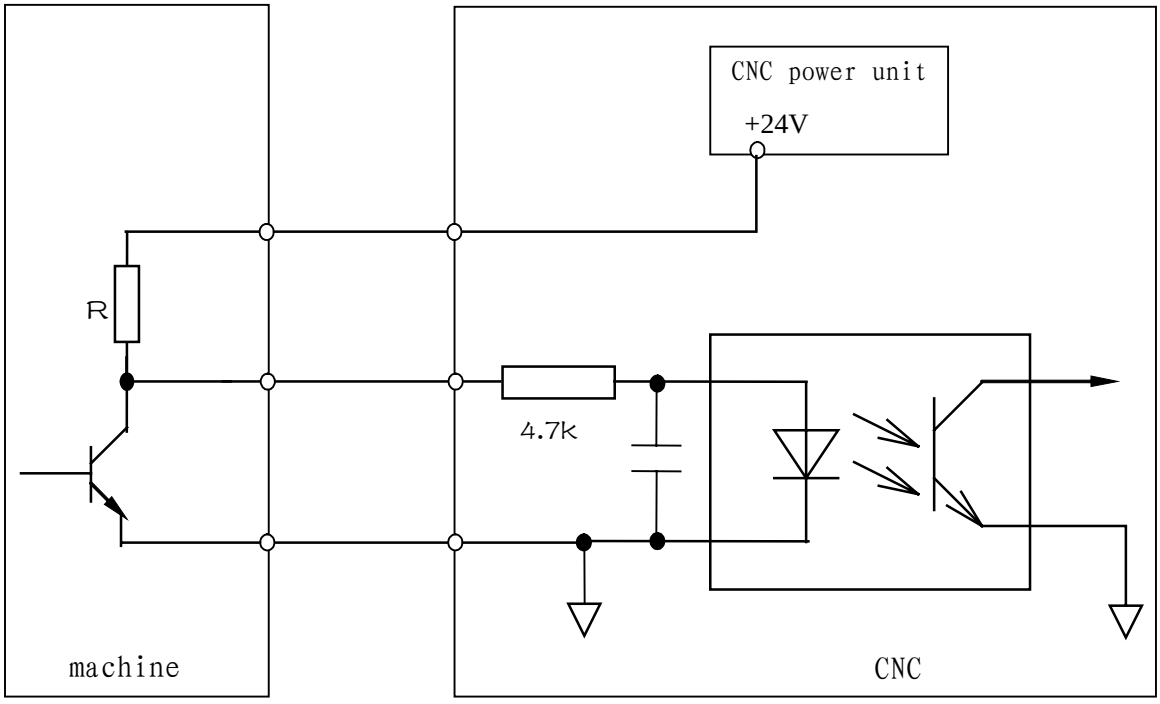


Fig. 4.1.2b

4.2 DESCRIPTION OF OUTPUT SIGNAL INTERFACE

DC output signal is used to drive relay and indicator lamp on machine tool side. The output circuit of this system is Darlington output.

4.2.1 Specifications of Darlington Output

(1) Output on: the maximum load current, including instantaneous current 200MA or less
(2) Output on: saturation voltage(at 200MA), the maximum value is 1.6V, typical value is 1V

(3) Output OFF: max. support voltage (including instantaneous voltage) is 24V＋20％ or less

(4) Output OFF: leakage current is 100μA or less.

4.2.2 Output Drive Relay Circuit

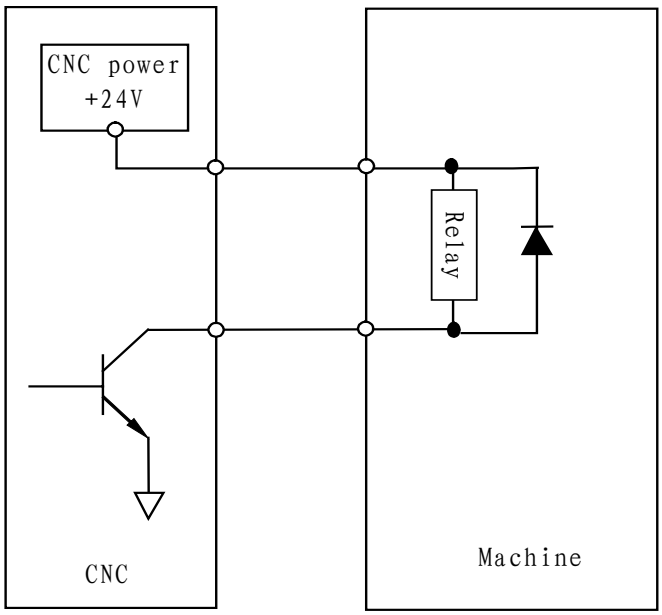


Fig. 4.2.2

Note：	**When an inductive load such as a relay is connected with the machine tool, a spark killer must be inserted as close to the load (within 20 cm) as possible. When a capacitive load is connected with the machine tool, a current limiting resistor must be connected in series. Loads must always be below the rated current and voltage including instantaneous values.**

4.2.3 Output Driver Indicator Lamp

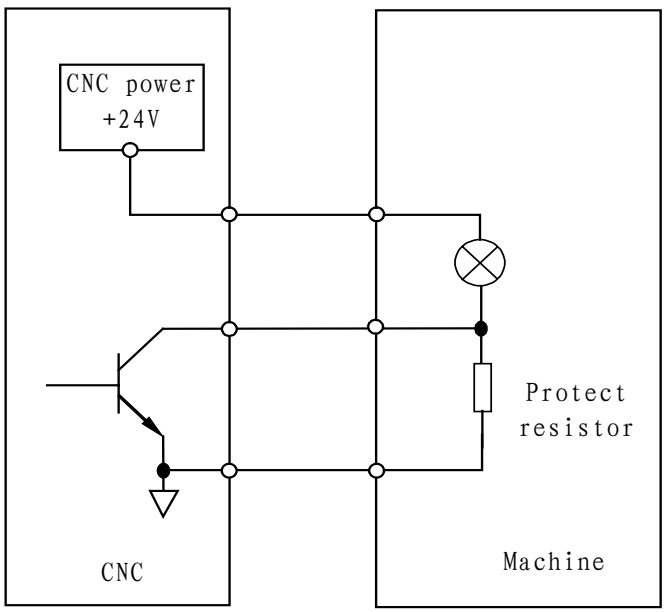


Fig. 4.2.3

Note： When transistor output lights the indicator lamp directly, shock current may be generate, which may be damage transistor easily. Therefore, a protection circuit must be inserted as shown above. Loads must always be below the rated current and voltage including instantaneous values.

4.3 INPUT/OUTPUT SIGNAL LIST

4.3.1 Input Signal Diagnose List

Bit No.:	7	6	5	4	3	2	1	0
Diagnose No. 000			*DECX	QPI				
Socket Pin No.			XS50:21	XS50:8				

Bit No.:	7	6	5	4	3	2	1	0
Diagnose No. 001			*DECZ	*ESP1	T04	T03	T02	T01
Socket Pin No.			XS50:20	XS50:7	XS50:19	XS50:6	XS50:18	XS50:5

Bit No.:	7	6	5	4	3	2	1	0
Diagnose No. 002					*ESP2	*SPL	*SPK	ST
Socket Pin No.					XS2:5	XS2:4	XS2:3	XS2:2

Bit No.:	7	6	5	4	3	2	1	0
Diagnose No. 003	X37	X36	X35	X34	X33	X32	X31	X30
Socket Pin No.	XS54:4	XS54:12	XS54:3	XS54:11	XS54:2	XS54:10	XS54:1	XS54:9
Function of input signal	*0V8	*0V4	*0V2	*0V1	T8	T7	T6	T5
					M93I	M91I	M23I	M21I
	/*LMZ	/*LPZ	/*LMX	/*LPX	*LMZ	*LPZ	*LMX	*LPX
	QPJI	QPSI					TWI	

4.3.2 Output Signal Diagnose List

Bit No.:	7	6	5	4	3	2	1	0
Diagnose No. 004	SPZD	TL−	TL+	Y14	M08	Y12	M04	M03
Socket Pin No.	XS50:1	XS50:14	XS50:2	XS50:15	XS50:3	XS50:16	XS50:4	XS50:17

Bit No1.:	7	6	5	4	3	2	1	0
Diagnose No. 005	QPJ	QPS	Y25	ESPO	S04	S03	S02	S01
Socket Pin No.	XS57:3	XS57:5	XS57:6	XS57:8	XS57:7	XS57:4	XS57:2	XS57:1

4.4 Input/Output Signal Arrangement at Socket XS50 and XS54

XS50:DB25F

1	SPZD

		14	TL-
2	TL+	15	Y14
3	M08	16	Y12
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 signal arrangementat socket XS54

XS54:DB15M

1	T06 (X31)	9	T05 (X30)
2	T08 (X33)	10	T07 (X32)
3	*0V2 (X35)	11	*0V1 (X34)
4	*0V8 (X37)	12	*0V4 (X36)
5	0V	13	+24V
6	0V	14	+24V
7	0V	15	+24V
8	0V		

Additional output signal arrangementat socket XS57

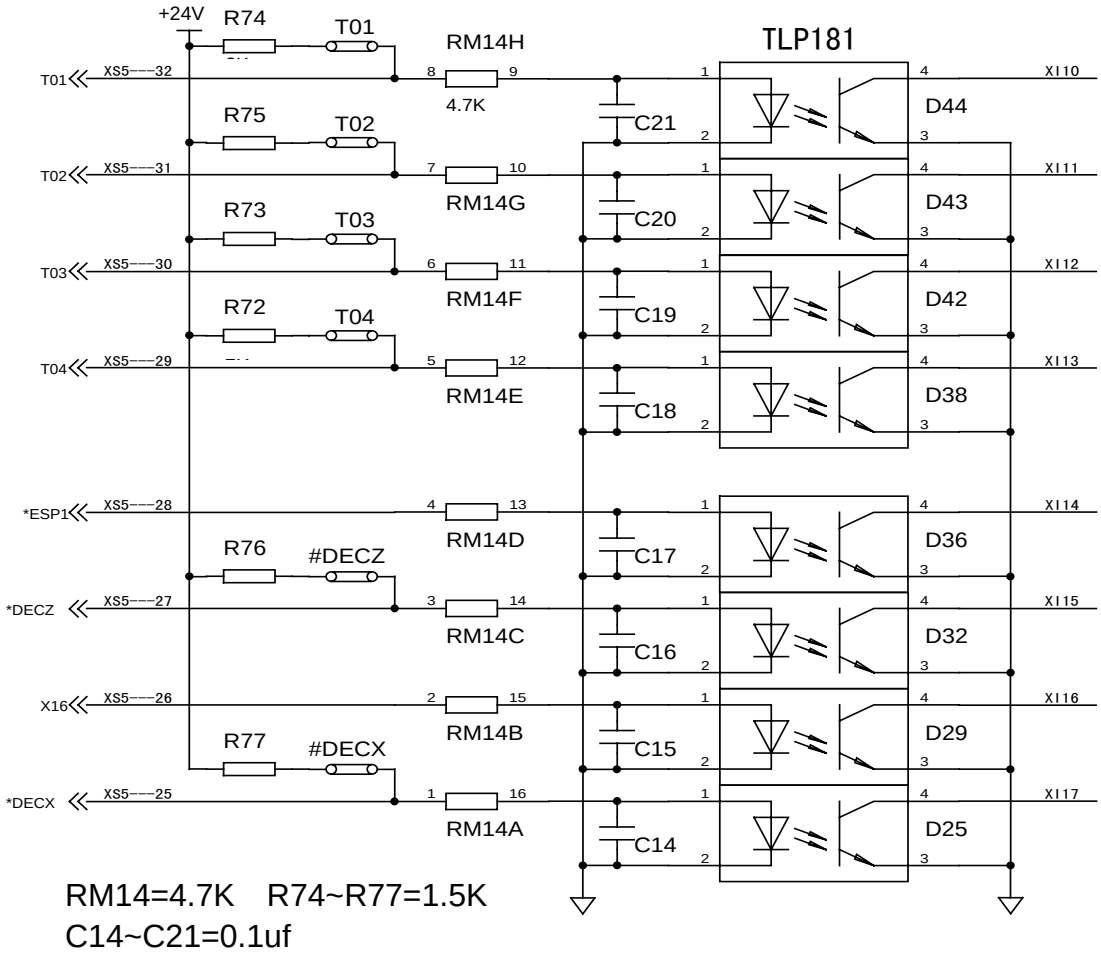
XS57:DB15M

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		

4.5 Input/Output Signal circuit

4.5.1 Input signal circuit

DI at socket XS50



RM14=4.7K　R74~R77=1.5K
C14~C21=0.1uf

Fig. 4.5.1a

DI at socket XS54

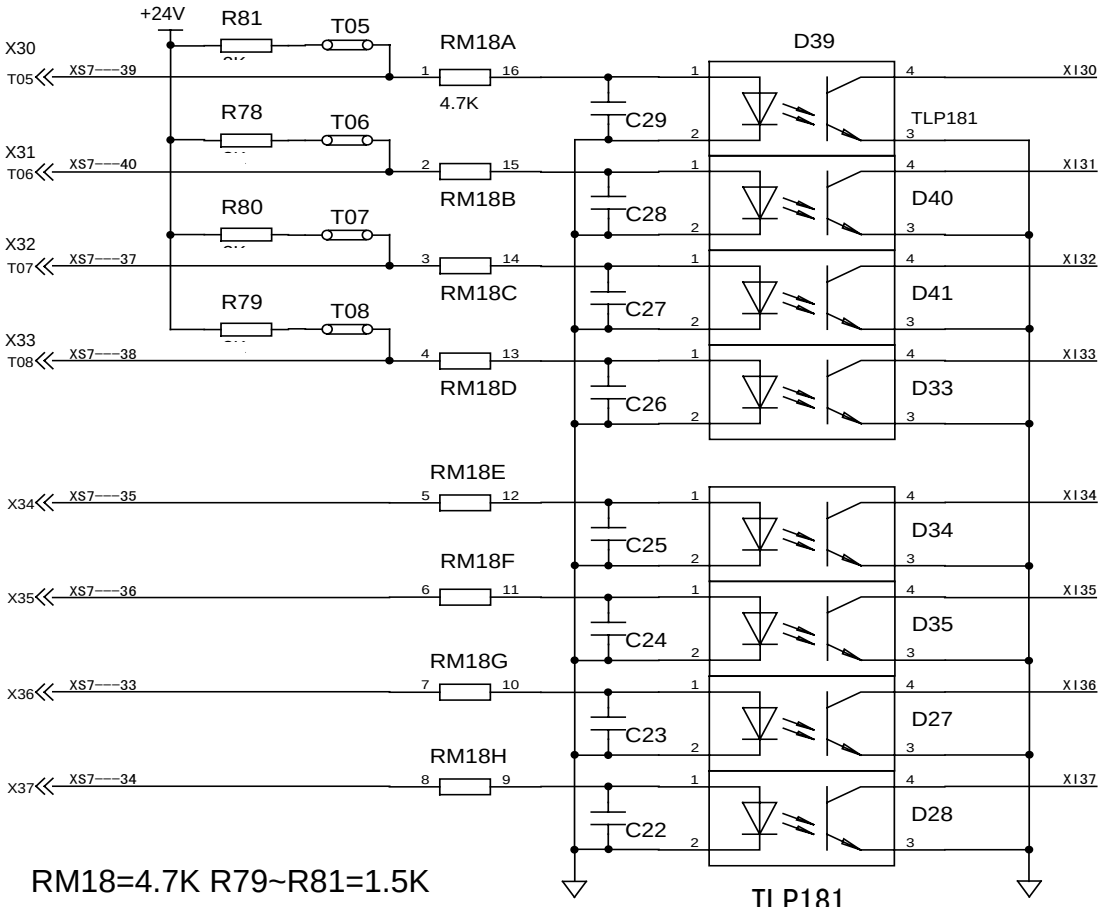


Fig. 4.5.1b

4.5.2 Output signal circuit

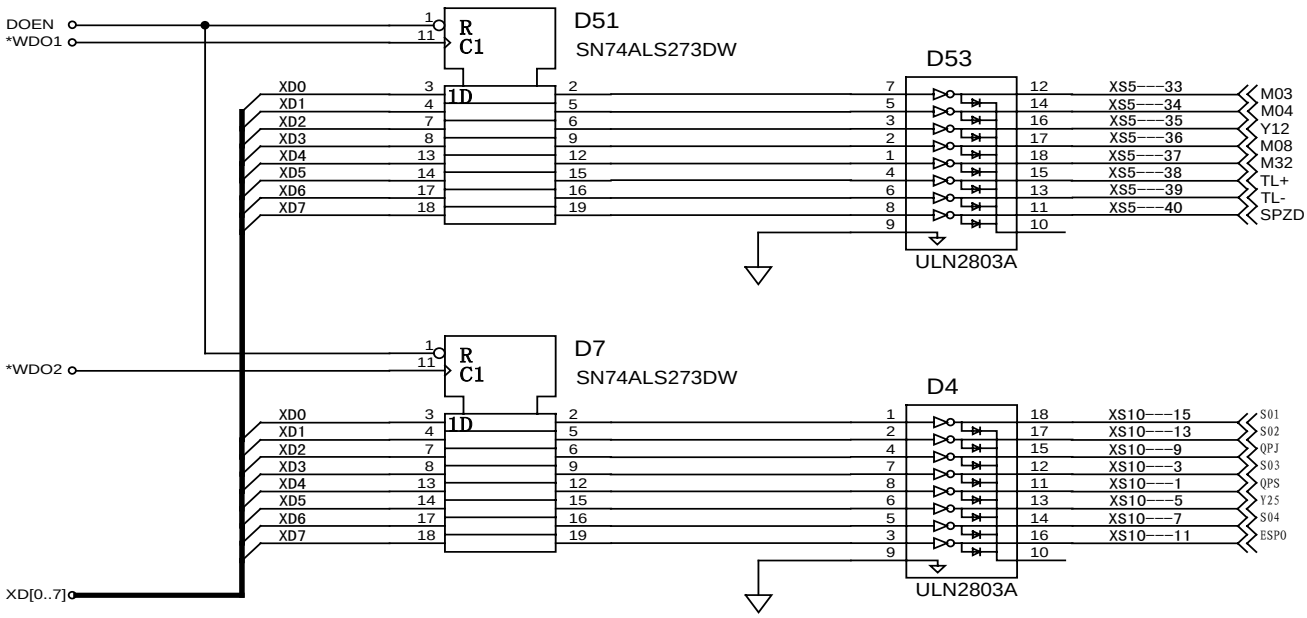


Fig. 4.5.2

4.6 Description of Input and Output Signal

4.6.1 Input Signal

(1) *DECX and *DECZ：deceleration switch signal

These signals are used to return reference point, and they are normal-closed contact. The procedure of reference point return is as follows.

The movable member of a machine tool moves at a rapid traverse by selecting manual mode, pressing the zero return key and by pressing a jog feed button to move the movable member in the direction of the reference point. By pressing the deceleration limit switch for one of the reference return deceleration signals (*DECX and *DECZ), the feedrate decelerates and after completion of deceleration the movable member continues moving at a constant low feedrate. When the deceleration limit switch is switched back to its original position and the CNC system will check one revolution signal or magne-switch signal (PC signal) of pulse coder. If the signal is from high voltage level to low voltage level (checking the descent trim of PC signal), feeding stops. The operation of reference point return completes. Jog feed will continuously be invalid during zero return .

The connection diagram is as follows：

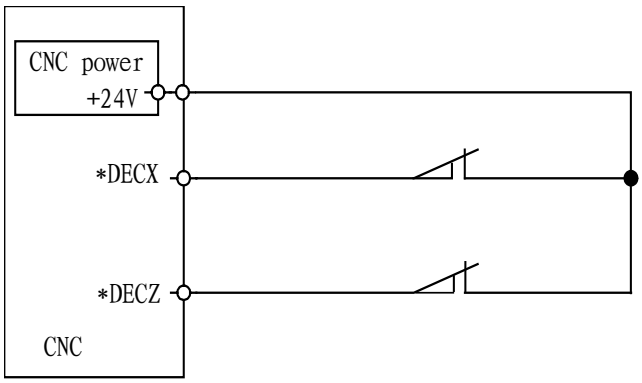


Fig. 4.6.1

(2) *ESP1 Emergency stop signal

This is a normal-closed contact signal. The control unit resets and machine tool comes to a halt when the contact of *ESP opens.

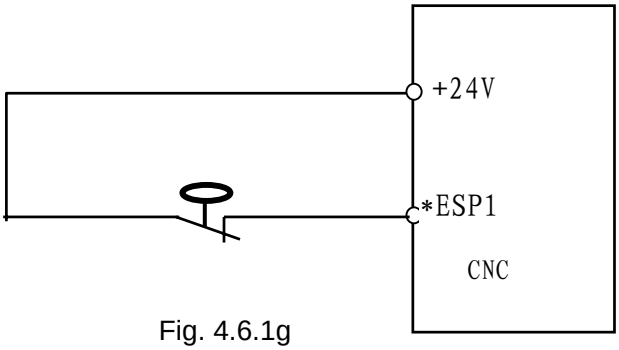


Fig. 4.6.1g

(3) *SPK：Feedhold signal

Used to hold the feed.

(4) ST：Cycle start

(5) *SPL：Spindle hold signal

(6) *OV8～*OV1

The override switch from the additional panel.

(7) /*LMZ、/*LPZ、/*LMX、/*LPX、*LMZ、*LPZ、*LMX、*LPX

The tool travel limit switch.

Related parameter

0	0	1

	MOT						

MOT 0: the travel limit switch is effective； 1: the travel limit switch is ineffective

Not：MOT has no effect on soft travel limit.

0	4	1

							LPMH

LPMH 0: the least four bit of DGN003 are specified as travel limit switch.

1: the most four bit of DGN003 are specified as travel limit switch.

The signal arrangement at socket

Diagnose No: 003	/*LMZ	/*LPZ	/*LMX	/*LPX	*LMZ	*LPZ	*LMX	*LPX
Socket Pin	XS54:4	XS54:12	XS54:3	XS54:11	XS54:2	XS54:10	XS54:1	XS54:9

Note： The travel limit switch is nomal-close.

4.6.2 Output Signal

This CNC system output signals are provided by Darlington. When output is effective, the corresponding Darlington will be conducting. The common terminal of overall output signal is +24V.

（1） TL＋，TL－The output signal to control the rotation of the tool carrier.

TL＋ the tool carrier rotate CW，TL－the tool carrier rotate CCW.

（2） M03，M04，M05 spindle control signal

M03: spindle rotate in clockwise，M04: spindle rotate in clockwise，

M05: spindle stop

（3） M08，M09 coolant control

M08: coolant open，M09: coolant close

（4） M32，M33 lubricant control

M32: lubricant open， M33: lubricant close.

（5） Spindle brake signal SPZD

Time charts is as below：

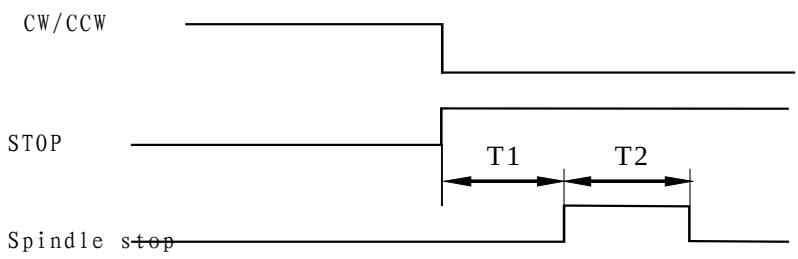


Fig. 4.6.2

T1：Specified spindle stop instruction (auto or manual) when it is rotating. Spindle stop signal is transmitted after delay time T1. Parameter T1 is set as 500ms.

T2：spindle brake time，set by parameter P040.

（6） Emergency stop output ESPO

ESPO is output if parameter M210/P041 is set 0 and the CNC emergency stop occurs or an driver alarm is detect.

（7） S01～S04

Control the spindle speed.

（8） M21O、M23O

Special M code.

V APPENDIX

APPENDIX 1: SPECIFICATION TABLE

Item	Name	Specification
Control axes	Control axes	2 axes (X, Z)
	Simultaneously controlled axes	2 axes
Command	Least command unit	0.001mm
	Least increment unit	0.001mm
	Max. command	±9999.999 mm
Feed	Rapid traverse rate	X:3 m/min，Z:6m/min（max.）
	Feed rate range mm/min	1～3000 mm/min
	mm/r(Decoder of 1024 pulse/r is required)	0.0001～500.0000 mm/r
	Lead of thread	0.0001～500.0000 mm
	Automatic acceleration/deceleration	Provided
	Feed rate override	0～150%
Manual	Manual continuous feed	One axis simultaneously , ×1,×10,×100 rate override
	Handle	Provided
Interpolation	Positioning , Linear Interpolation , Circular Interpolation	G00,G01,G02/G03
Test	Dry run, Single Block	Provided
Canned Cycle	Outer diameter/internal diameter cutting canned cycle	G90
	the thread cutting canned cycle	G92
	the end face turning canned cycle	G94
MULTIPLE REPETITIVE CYCLE	Stock Removal in Turning	G71
	Stock Removal in Facing	G72
	Pattern Repeating	G73
	End Face Peck Drilling Cycle	G74
	Outer Ddiameter Drilling Cycle	G75
	Multiple Thread Cutting Cycle	G76
Coordinate and Dwell	Dwell(s)	G04
	Work coordinate system setting	G50
	Auto coordinate system setting	Provided
Mode	MDI,AUTO,JOG,STEP,EDIT	Provided
Safety	Stored stroke limit check	Provided
	Enable/disable Stored stroke limit check	Provided
	Emergency stop	Provided

Item	Name	Specification
Program Edit and storage	Capacity of program memory	32K size and 63 program
	Program editing	Insert,Alter and Delete
	Program number，Sequence number，Address，Word search	Provided
	Decimal point inpu	Provided
	Flash memory	Provided ，6 region
Display	320×240, 6 inch LCD(1TB)	192×64 LCD（**1TA**）
	Position,Program,Offset,Alarm,Test,Diagnose and Parameter	Provided
	Graph	Provided
M，S，T	DI/DO：20 / 8	DI:4 from panel, 16 from machine
	Miscellaneous function	Mxx
	Spindle function	Sxx
	Analogue spindle output （Sxxxx），Constant surface speed	Provided
	Tool selection	T01～08
	Check the tool carrier periodically	Provided
Offset	Tool compensation memory	±6 bit, 8 group
	increment input	Provided
	absolute input	Provided
	Backlash	Provided
Protect switch	Program protect switch, Parameter protect switch,	Provided
Other function	Circular radius R specifying	Provided
	Command multiply for each axis.	Provided
	Retain the absolute coordinate at power off.	Provided
	Selection of tail tool carrier	Provided
	The program can start from any block.	Provided
Extension	G00 traverse rate:12 m/min ，G01 feedrate:8m/min	Provided
Selection	Additional operation panel（K1TBII）	Provided

APPENDIX2 PARAMETERS

Parameters not mentioned in this table must be set by 0.

0	0	1	MSPL	MOT	MESP	SINC	CPF4	CPF3	CPF2	CPF1

MSPL 1:The followint signal from panel are ineffective. *ESP2，*SPL，*SPK.

MOT 1:The travel limit switch is ineffective.

MESP 1: *ESP1 is ineffective.

SINC 1:The most significant two increment of Step and Handle is unavailable. 0.1，1

CPF4，3，2，1 Pulse frequency of backlash compensation and pitch error compensation (common for all axis).

compensation frequency = （setting value +1）Kpps Setting amount = 0 ～15.

CPF4,3,2,1 will not be effective until the power is turned off.

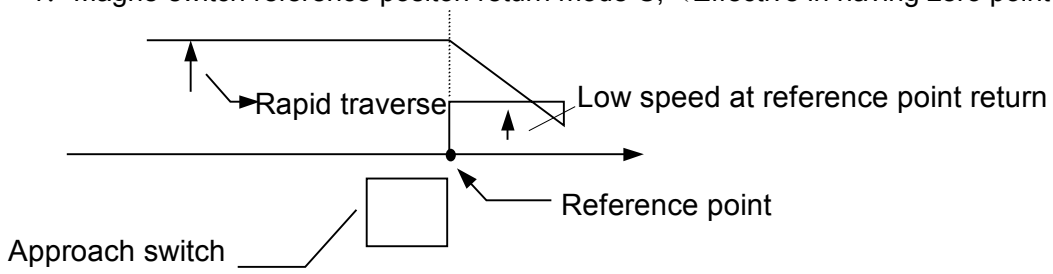
0	0	2	ZRSZ	ZRSX	ZCZ	ZCX	DALZ	DALX	DIRZ	DIRX

ZRSZ，X 1: The axis has reference point （Reference point return B）

0: No reference point（Reference point return A）

ZCX，Z 0：Deceleration switch and zero signal is essential in zero point returning.

1：Magne-switch reference positon return mode C,（Effective in having zero point）



When mode C is effective，parameter ZMZ /P004 should be set simultaneously.

DALZ~X：Select the voltage level of the driver alarm signal for each axis.

DIRZ~X：Select the motor rotation direction for each axis.

0	0	3	BDEC	BD8	RVDL	SMZ	KSGN	ZNIK	TSGN	ABOT

BDEC 0：Set the fixed frequency (Setting by CPF4,3,2,1and BD8) in backlash compensation A mode.

1：Set the frequency by acceleration/deceleration method(CPF4,3,2,1 and BD8 is ineffective)

BD8 0：Backlash compensation is executed by the frequency set by P001

1：Backlash compensation is executed by the 1/8 frequency set by P001

RVDL 0：Direction and pulse signals are output at same time when the axis direction is changed.

1： Direction signal exceeds pulse for a time when the axis direction is changed.

SMZ 0：Speed control at the corner of block refer to “Feed Function” of Program manual.

1：The next block can be executed after the speed of all motion command decelerates to zero

KSGN 0：Direction signal doesn’t keep on when the axis moves in the negative direction.

1：Direction signal keeps on when the axis moves in the negative direction.

ZNLK 0：During the reference point return, the axis moving key can't keep on.

1：During the reference point return, the axis moving key can keep on. After returning the point, stop movement. In this course, we can stop the movement by pressing "Reference return "or "RESET".

TSGN 0 : The high voltage level signal indicates the tool in position.

1 : The low voltage level signal indicates the tool in position.

ABOT 0 : The absolute coordinate is retained during power off.

1 : The absolute coordinate is not retained during power off.

0 0 4	SANG	HPG	XRC	SOVI	MZRZ	MZRX	ZMZ	ZMX

SANG 1：Select the analog spindle function

HPG 1：Handle is effective

XRC 0：Diameter programing for X axis

1：Radius programing for X axis

SOVI 1：Select external switch as Feedrate override,the one from the panel is ineffective.

MZRZ～X：Reference point return direction of each axis

0：Reference point return is performed in the negative direction.

1：Reference point return is performed in the positive direction.

ZMX ZMZ：Reference point return direction of each axis and the initial backlash at power on.

1：Reference point return and the backlash are performed in the negative direction.

0：Reference point return and the backlash are performed in the positive direction.

0 0 5	CMRX

0 0 6	CMRZ

CMRX CMRZ： Command multiply for each axis.

Setting range：1～127

0 0 7	CMDX

0 0 8	CMDZ

CMDX CMDZ：Command pulse multiplying factor denominator for each axis.

Setting range：1～127

0 0 9	RPDFX

0 1 0	RPDFZ

Setting amount: 30～15000 Unit: mm/min(MM output).

30～6000 Unit: 0.1inch/min(inch output).

RPDFX，Z : Rapid traverse of each axis.

Setting amount: X axis：1～3000 mm/min（Standard）,1～6000 mm/min,（Extend）

Z axis：1～6000 mm/min（Standard）,1～12000 mm/min（Extend）

0 1 1	BKLX

0	1	2	BKLZ

BKLX BKLZ Backlash amount of each axis

Setting range: 0～2000 Unit: 0.001mm

0	1	3	PRSX

0	1	4	PRSZ

PRSX PRSZ: It sets the coordinates for automatic coordinate system setting after zero return is executed.

Setting amount: 0～±9999999

0	1	5	LT1X1

0	1	6	LT1Z1

0	1	7	LT1X2

0	1	8	LT1Z2

LT1X1/Z1：The stored travel limit for each axis in positive direction.

LT1X2/Z2：The stored travel limit for each axis in negative direction.（Set diameter value for X axis if programing in diameter.）

Setting range： 0～±9999999(Unit:0.001mm)

Setting the distance from zero point to stroke limit. Out of the setting area is called inhibit area. We often set stoke limit in Max. stroke. Over travel alarm will occurr if the machine move in inhibit area.

An allowance must be calculated in a moving interval. It is one fifth of the rapid traverse (Unit：mm). For example, if rapid traverse= 3m/min, allowance= 3×1/5 = 0.6mm. There is an allowance between the CNC position and the machine's because of step losses, so that zero point return must be carried out before machining. Otherwise, over-travel checking point will include this allowance.

0	1	9	LINTX

0	2	0	LINTZ

LINTX LINTZ Time constant of linear acceleration/deceleration in rapid traverse.

Setting amount： 8～4000 ms

0	2	1	FEEDT

FEEDT Time constant of exponential acceleration/deceleration in feed.

Setting amount：0 ～4000 Unit：ms

When set by zero, the function of exponential acceleration/deceleration is ineffective

0	2	2	FEDFL

FEDFL The lower limit speed on exponential acceleration/deceleration in feed.

Setting amount：0 ～3000 Unit：mm/min (mm output)

This parameter is set to 40 in general.

0 2 3	THRDT

THRDT Time constant of exponential acceleration/deceleration along X axis in thread cutting (G92) .

Setting amount： 0～4000mm

0 2 4	THDFL

THDFT **The lower limit speed on exponential acceleration/deceleration in thread cutting** (G92). **(Common to all axis)**

Setting amount：6～3000 mm/min

0 2 5	FEDMX

FEDMX The upper limit speed of cutting feed (Common to all axis).

Setting amount：6～3000 mm/min（Standard）。0～8000 mm/min（Extend）

0 2 6	RPDFL

RPDFL The least speed of rapid traverse override (FO)(Common to all axis),

Setting amount：6～3000Unit：mm/min(mm output)

0 2 7	ZRNFL

ZRNFL Low feed speed at reference point return(Common to all axis)。

Setting amount：6～3000 Unit：mm/min

0 2 8	JOGFL

JOGFL The least speed of manual feed .(FL)。

Setting amount：6～3000 Unit：mm/min

0 2 9	SEQNIC

SEQINC The increment of sequence number increasing

Setting amount：0～9999

If SEQINC＝0， the sequence number can not be insert automatically;

If SEQINC≠0， the sequence number can be insert automatically ;

0 3 0	WLKTME

WLKTME：Delay time of signal

Standard value is 2. When the power on, the system check this parameter automatically. If it is more than 15 or less than 2,it will be set by 2.

0 3 1	GRMAX1

0 3 2	GRMAX2

GRMAX1～ GRMAX2 Data adjustment for gain in spindle analog output. The value should be equal to the actual spindle speed when the anolog signal is 10V .

Setting amount: 1～9999 rpm

0 3 3	LOWSP

LOWSP The lower spindle speed limit during the constant surface speed control. (G96)
Setting amount: 1～9999 rpm

0 3 4	TCTMX

TCTMX：The maximum delay time for tool selectoin
Setting amount: 1～100000 mm

0 3 5	MTIME

MTIME： The time delayed by M code
Setting amount: 1～4080 ms

0 3 6	CKTDI	QSEL	AGER	QPSL	RVX		SM02	SM01

CKTDI 1: To scans and checks the tool frame signal at certain interval .
QSEL 1：The multiple cycle is effective.
AGER 0：Always 0.
QPSL 1：The chuck function is available
RVX 1: The tail tool carrier is select.
SM02,SM01: Specify the meaning of Y14,Y12.

0 3 7	T1

T1： The delay time from tool carrier positive revolution stop to its reversal locked singal output.
Setting amount: 1～4080 ms

0 3 8	TLOCK

The pulse width of tool carrier reversal locked singal. Setting amount: 1～4080 ms

0 3 9	TOOLNO

TOOLNO Set the total tool number
Setting amount: 1～8

0 4 0	SPZDTIME

The spindle brake time. Setting amount: 1～32640 ms

0 4 1	QPLS	QPM3	ZG92L	G93N	TWSL	M23O	M21O	LPMH

QPLS 0：The chuck control signal is high/low voltage level.
1：The chuck control signal is pulse and the width of pulse is set by parameter P051.
QPM3：0：Before actuating the spindle,the chuck in position signal is checked.If the signal is not detected, the program will be intermited and an alarm (NO. 015) is output.
1：No check is done.
ZG92L: 0: During thread cutting (G92/G76), the speed control of Z axis is exponential acceleration/deceleration.
1: During thread cutting (G92/G76), the speed control of Z axis is linear acceleration/deceleration.

G93N: 0：During G93 execution ， no acceleration/deceleration control is adopted.

1：During G93 execution ， exponential acceleration/deceleration control is adopted.

TWSL: 1: The tail is available

M23O：1：Set the BIT5 of Diagnose byte data No.005 as the status of M23 output.

M21O：0：Set the BIT4 of Diagnose byte data No.005 as the status of ESP output.

1：Set the BIT4 of Diagnose byte data No.005 as the status of M21 output.

LPMH：0：Set the low 4 bit of Diagnose byte data No.003 as the status of travel limit switch.

1：Set the high 4 bit of Diagnose byte data No.003 as the status of travel limit switch.

0 4 2	OFMD2						XG92L	XG92R

OFMD2 1: Offset mode 2，The tool offset can only be set in absolute input.

XG92L 0: During thread cutting (G92/G76), the exponential acceleration/deceleration is effective for X axis and the time constant is the same as normal.

1: During thread cutting (G92/G76), the linear acceleration/deceleration is effective for X axis and the time constant is set by parameter P057.

XG92R 0: During chamfering in thread cutting (G92/G76),the tool move along X axis at the cutting speed .

1: During chamfering in thread cutting (G92/G76),the tool move along X axis at the rapid traverse rate.

0 4 3	QPIN							

QPIN 1: The input signal for chuck position is available.

0 4 5	STIME1

STIME1：Delay time of S code，0～4080ms

0 4 6	STIME2

STIME2：Delay time of S code，0～4080ms

0 5 1	QPLSTIME

QPLSTIME：The width of chuck command pulse. (ms)

0 5 7/8	G92LINTX/Z

G92LINTX/Z: time constant for linear acceleration/deceleration during thread cutting.

Parameter used in multiple repetitive cycle (G70～G76)

The content of the second parameter page can switched by pressing SHIFT key.The meaning of P021～P031 in SHIFT state is described .

P’021：MRCCD Depth of cut for multiple repetitive cycle (G71,G72)

P’022：MRCDT Escaping amount for multiple repetitive cycle (G71,G72)

P’023：PECSCX Distance of relief in the X axis direction for multiple repetitive cycle (G73)

P’024：PECSCZ Distance of relief in the Z axis direction for multiple repetitive cycle (G73)

P’025：PATIM the repetitive count for multiple repetitive cycle (G73)

P’026：GROVE Escaping amount for multiple repetitive cycle (G74)

P’027：THRPT Repetitive count in finishing for multiple repetitive cycle (G76)

P’028：THDCH Chamfering amount for thread cutting (G92,G76).

P’029：THANG Angle of tool tip for multiple repetitive cycle (G76)

P’030：THCLM Minimum cutting depth for multiple repetitive cycle (G76)

P’031：THDFN Finishing allowance for multiple repetitive cycle (G76)

These parameters can be set by MDI or by program.

The address P contained in G92 command block change the P'28 also. If the value of P028 is available, the address P in G92 command block can be omitted.

Standard value and meaning of **K1TBⅡ**

No.	Standard value	Meaning
1	01000000	Bit parameter
2	00000000	Bit parameter
3	11101110	Bit parameter
4	00000000	Bit parameter
5	1	CMR for X-axis
6	1	CMR for Z-axis
7	10	CMD for X-axis
8	10	CMD for Z-axis
9	3000/4000	Rapid traverse rate for X-axis
10	4000	Rapid traverse rate for Z-axis
11	0	Backlash along X axis
12	0	Backlash along Z axis
13	0	The coordinate value for X axis at the reference position.
14	0	The coordinate value for Z axis at the reference position
15	9999999	The stored travel limit for X axis in positive direction
16	9999999	The stored travel limit for Z axis in positive direction
17	-9999999	The stored travel limit for X axis in negative direction
18	-9999999	The stored travel limit for Z axis in negative direction
19	300	Linear acc/dec constant for X axis
20	300	Linear acc/dec constant for Z axis
21	100	Exponential acc/dec constant during cutting or Jog
22	100	The lower limit speed on exponential acc/dec in feed
23	100	Time constant of exponential acc/dec along X axis in thread cutting
24	350	The lower limit speed on exponential acc/dec in thread cutting
25	3000/4000	The upper limit speed of cutting feed
26	50	The least speed of rapid traverse override (FO)
27	200	Low feed speed at reference point return(FL)
28	40	The least speed of manual feed
29	10	The increment of sequence number
30	2	Signal filter time
31	9999	Analogue spindle output adjust
32	9999	Analogue spindle output adjust
33	99	The spindle speed lower limit during constant surface speed control.
34	80000	The max. delay time for tool selection

No.	Standard value	Meaning
35	1024	The delay time for M code
36	01000010	Bit parameter
37	96	The delay time from tool carrier positive revolution stop to its reversal locked singal output
38	976	The pulse width of tool carrier reversal locked singal.
39	4	The total tool number
40	2000	Spindle brake time
41	01110000	Bit parameter
42	10000010	Bit parameter
43	00000000	Bit parameter
44	00000000	Bit parameter
45	16	The delay time for S code
46	16	The delay time for S code
47	0	No used
48	0	No used
49	0	No used
50	0	No used
51	32	The pulse width of chuck
52	0	No used
53	0	No used
54	0	No used
55	0	No used
56	0	No used
57	150	Linear acc/dec constant for X axis for G92/G76
58	150	Linear acc/dec constant for Z axis for G92/G76
59	0	No used
60	0	No used

APPENDIX3 CNC STATUS DIAGNOSE MESSAGE

1 NORMAL DIAGNOSE MESSAGE

1.1 Input signal

0	0	0			*DECX	QPI				

QPI: foot switch to control chuck

0	0	1			*DECZ	*ESP1	T04	T03	T02	T01
0	0	2					*ESP2	*SPL	*SPK	ST
0	0	3	X37	X36	X35	X34	X33	X32	X31	X30
Override/Tool			*OV8	*OV4	*OV2	*OV1	T08	T07	T06	T05
Program							M93I	M91I	M23I	M21I
Travel limit SW			/*LMZ	/*LPZ	/*LMX	/*LPX	*LMZ	*LPZ	*LMX	*LPX
Tail control									TWI	

1.2 Output signal

0	0	4	SPZD	TL－	TL＋	Y14	M08	Y12	M04	M03
0	0	5	QPJ	QPS	Y25	ESPO	S4	S3	S2	S1
Program					M23O	M21O				
Tail control					TWJ	TWT				

QPJ: Chuck close QPS: Chuck open

TWJ:Tail go forward TWT:Tail go backward

1.3 Status signal

0	0	6		CSCT	CITL	COVZ	CINP	CDWL	CMTN	CFIN
0	0	7	STP	REST	EMS		RSTB			CSU

1.4 MDI key signal

0	0	8	7	6	5	4	3	2	1	0
0	0	9	*SPK	RST	W	P	T	M	9	8
0	1	0		CAN	EOB	SHT		DEL	INS	ALT
0	1	1					PGU	PGD	CRU	CRD
0	1	2	TOL	COL		SPL	F0		MOD	DSP
0	1	3				RT	＋Z	－Z	＋X	－X

0	1	4	RERRX
0	1	5	RERRZ

RERRX the accumulated error axis or pulse of current step along X

RERRZ the accumulated error axis or pulse of current step along Z

Note：Press〖SHIFT〗to select one of the two meaning.

2 Special DIAGNOSE MESSAGE

When the nomal diagnose message is on the screeen , pressing key INSERT and key 1 simultaneously switch the screen to special diagnose message.

1 Diagnose No.0～1 is same as normal.

2 Signal for CNC

0	0	6			RFZ	RFX		PCS	PCZ	PCX
0	0	7			1	1			ALMZ	ALMX

3 Signal for NC

0	0	8	HX/RV1		*DECX		－X	＋X		
0	0	9	HZ/RV2		*DECZ		－Z	＋Z		
0	1	0	DRN				GR2	GR1		
0	1	1	MLK	MP2	MP1		SBK	BDT		
0	1	2	ZRN	*SSTP	SOR	SAR	FIN	ST	STLK	MIR1
0	1	3	ERS	RT	*SP	*ESP	*OV8	*OV4	*OV2	*OV1
0	1	4	PN8	PN4	PN2	PN1	KEY	MD4	MD2	MD1
0	1	5	CDZ	SMZ	AFL	OVC		SOVC	SOVB	SOVA

APPENDIX4: ALARM LIST

1 Operation alarm(P/S Alarm)

Number	Content	Remarks
000	Re-apply 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 numeral, the sign (-) or a decimal point was input without an address at the beginning of a block.	
005	The address was not followed by appropriate 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 search.	
061	The address P or Q is not specified in command G70/G71/G72/G73 .	
062	（1）Depth of cut for multiple repetitive cycle (G71,G72) is zero or negative. （2）the repetitive count for multiple repetitive cycle (G73) is zero or negative. （3）The value of △I or △j for G74 or G75 is negative. （4）The value of △I or △j for G74 or G75 is not equal zero，but the value specified by U or W is not lager than zero. （5）The Escaping direction is specified but the value of △d is negative. （6）The cut depth for G76 is negative. （7）The minimum cutting depth is larger than the total cut depth for G76. （8）Angle of tool tip for G76 is not legal.	
063	The sequence nuber specifed in G70/G71/G72/G73 command can not 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 unallowed G code.	
067	In MDI mode, the command G70/G71/G72/G73 with P or Q is entered.	
068	The memory is over.	
071	The address searched for 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 block which including M98.	
077	The subprogram was over the nesting depth limit.	
078	The program number or the sequence number which was specified by address P in a block containing M98 was not found between M98 and M99.	
090	Try to excute reference position return before set it.	
101	The power was turned off while rewriting the contents of memory/program. When this alarm was occurred, you must turn off the power and turn on again.	

2 Over travel limit

Number	Contents	Remarks
+X	Over +X travel limit	
-X	Over -X travel limit	
+Z	Over +Z travel limit	
-Z	Over -Z travel limit	

3 Driver alarm

Number	Contents	Remarks
12	Driver device alarm for X-axis	
22	Driver device alarm for Z-axis	
13	The feedrate is too large for X axis. It is generated by CMR or CMD set incorrectly	
23	The feedrate is too large for Y axis. It is generated by CMR or CMD set incorrectly	

4 CNC alarm

Number	Contents	Remarks
02	CMOS error	
03	ROM parity	
06	WATCH DOG alarm	
07	CPU error (0,3,4,6 alarm)	
08	Illegal no mask interrupt	

Note：CNC alarm is often raised by unnormal +5V voltage .

5 External message alarm

Number	Contents	Remarks
01	M code alarm. Illegal M code was specified in program.	
02	S code alarm. Illegal S code was specified in program.	
03	T code alarm. Illegal T code was specified in program.	
05	The delay time for tool selection is over upper limit.	
06	M04 was incorrectly specified. While Spindle rotating CW(CCW), M04 (M03) was commanded without spindle stop.	
08	The parameter of total tool number is error.	

Ⅵ K2T CNC system

The programing in K2T CNC system is the same as in K1TBⅡ CNC system. However the operation is different and the functions is more stong. In this section , the difference is specified.

1 New function in K2T CNC system

- ▲ RS232 communication port；
- ▲ Extended Input/Output（selection）；
- ▲ The format of axis motion pulse can be set by parameter.

2 Installation Dimension of CNC Control unit

2.1 Panel

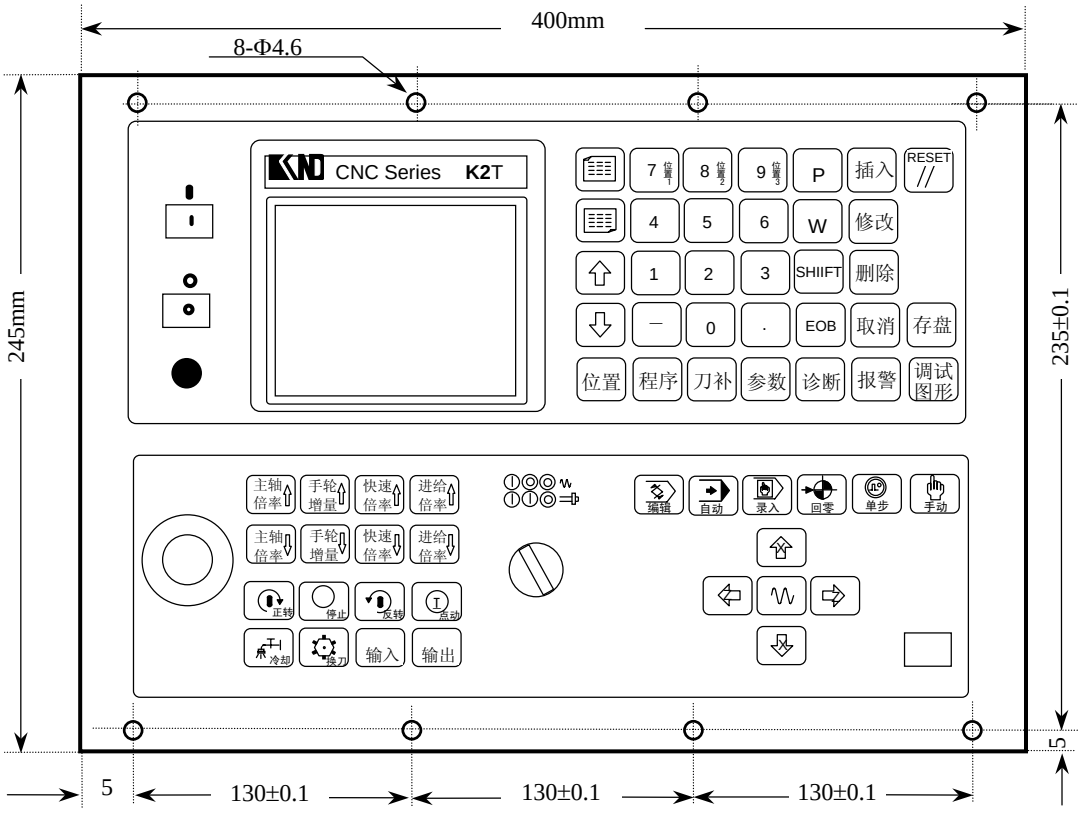


Fig. 2.1 Front view

2.2 Left view of K2T CNC system

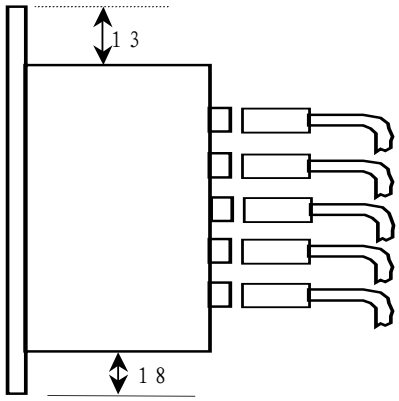


Fig. 2.2 Left view

2.3 Upside view of K2T CNC system

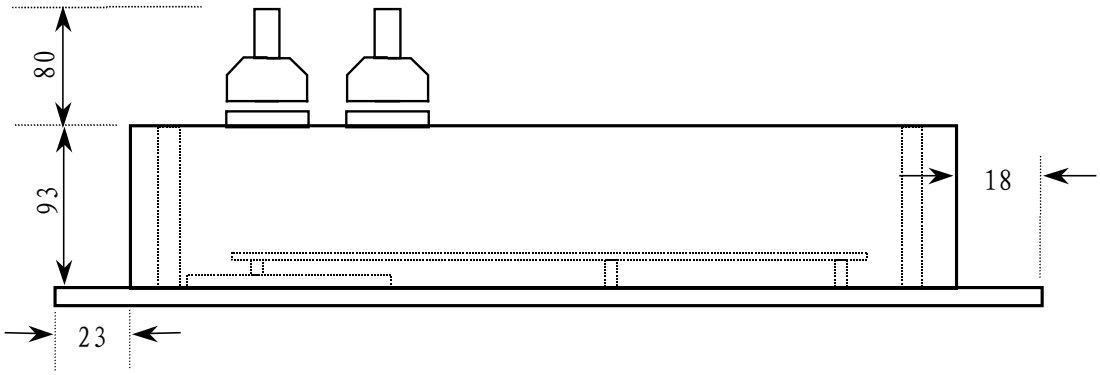
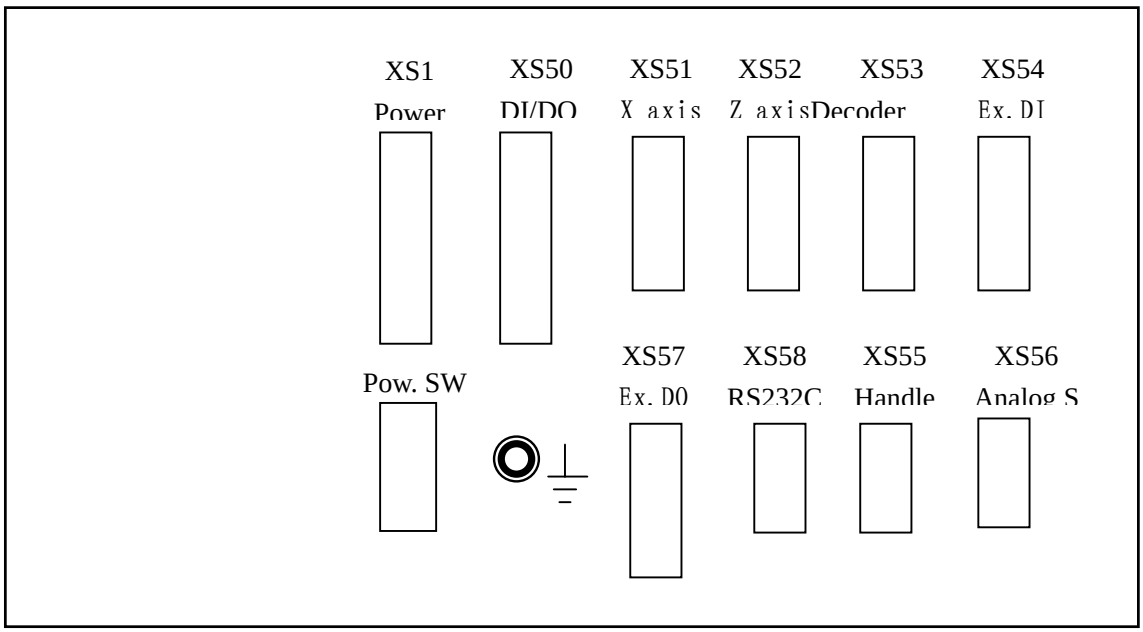


Fig. 2.3 Upside view

2.4 Back cover Diagram



3 Setting switch description

Switch on mani board

Name	Status		Explanation	Note
SA3	1，2	OFF	Default status	If 1,2 is set on,the system is reset.
	3，4	ON		
SA4		ON	Voltage of one rotation signal used for X returning reference position is +5V.	one rotation signal is fed by decoder
		OFF	Voltage of one rotation signal used for X returning reference position is +24V.	one rotation signal is fed by proximity switch
SA5		ON	Voltage of one rotation signal used for Z returning reference position is +5V.	one rotation signal is fed by decoder
		ON	Voltage of one rotation signal used for Z returning reference position is +24V.	one rotation signal is fed by proximity switch
SA6		1-2 ON	Voltage of VP is +24V	
		2-3 ON	Voltage of VP is +5V	
SA8 SA9 SA10		ON	The signal from spindle decoder is sigle polarity	SA9,SA10,SA11 should be set in the same status.
		OFF	The signal from spindle decoder is double polarity	
T01~T04		ON	The DI is pulled up internal	Refer to 4-3
		OFF	The DI is not pulled up internal	
T05~T08 #DECX #DECZ		ON	The DI is pulled up internal	
		OFF	The DI is not pulled up internal	

4 RS232 communication

4.1 Preparation

4.1.1 Connection of communication cable

（1） The communication cable should be connected with the CNC and PC power-off to avoid hardware damage.

（2） The communication cable diagram is shown in Fig. 4.1.1。

4.1.2 Parameter

0	0	4

			PUCH				

PUCH 1 : RS232 communication function is effective.

Note1：Turn off the power of CNC after this parameter is set.

Note2：Don’t set this parameter if RS232 is not available.

0	5	7	BAUTE

BAUTE:Set bps of RS232. The default value is 2400.

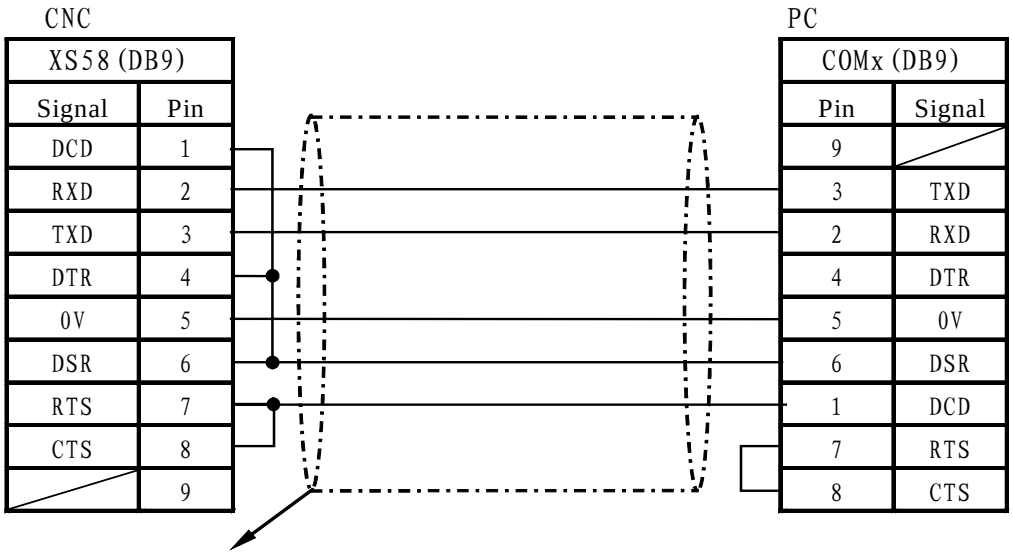


Fig. 4.1.1 Commu. cable

4.2 PROGRAM OUTPUT

Transmit the registered program in the memory to computer.

(1) Connect PC computer.
(2) Set the mode selector switch to EDIT.
(3) Press the [PRGRM] button to display program.
(4) Run communication software to be in wait mode.
(5) Press the address key O.
(6) Input the program number.
(7) Press the key [OUTPUT] to transmit the corresponding program to computer.

Note: Press the key [RESET] to stop output.

4.3 ALL PROGRAM OUTPUT

Transmit all registered program in the memory to computer.

(1) Connect PC computer and run KND communication software.
(2) Set the mode selector switch to EDIT.
(3) Press the [PRGRM] button to display program.
(4) Press the address key O.
(5) Press the keys –9999 and the key [OUTPUT].

4.4 Program input

(1) Turn on the program protect switch.
(2) Set the mode selector switch to EDIT.

(3) Press the [PRGRM] button to display program.
(4) Press the address key O and the program number.
(5) Connect PC computer and run KND communication software.
(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 containing Program number and no change is needed,the step (4) can be omitted.

4.5 FILE INCLUDING MANY PROGRAMS STORED INTO MEMORY

The content of program file is as follows.

```
O1111;
……;
……;
…….;
M30;
%
O2222;
……;
……;
M30;
%
O3333;
……;
……;
M30;
%
```

This operation is the same as 4.4. several programs store into memory with % end.

4.6 Program Comparation between in Memory and in Programmer

Turn off the program protect switch, this operation is the same as storing files into memory. Compare the program registered in memory and in programmer.

(1) Set the mode selector switch to EDIT or AUTO.
(2) Turn off the program protect switch.
(3) Connect the computer and run KND communication software to be in output mode.
(4) Press the key [PRGRM] to display the program on the LCD screen.
(5) Press the key [INPUT].
(6) There are several programs in the file, checking up to ER(%). “Comparation” is displayed on the screen.

Note:

1 Alarm P/S No.79 occurs and check stops when difference appears at the comparation.

2 Turn on the program protect switch and repeat the above operation, program is stored in memory but comparation can’t execute.

4.7 Parameter Output

Transmit the registered program in the memory to computer.

(1) Connect PC computer.
(2) Set the mode selector switch to EDIT.
(3) Press the [PARAM] button to display parameter.
(4) Run communication software to be in wait mode.
(5) Press the key [OUTPUT] to transmit the parameter to computer.

4.8 Parameter Input

(1) Turn on the parameter protect switch.
(2) Set the mode selector switch to EDIT.
(3) Press the [PARAM] button to display program.
(4) Press the [INPUT] button
(5) Connect PC computer and run KND communication software.
(6) Select the parameter file and set it to output status in the KND communication software. Then the parameter is transmitted from PC to CNC.

5 Additional DI/DO （Selection）

The K2T CNC system contains 20 DI and 16 DO by default. By selecting the additional DI/DO that contains 60 DI and 40 DO , The K2T CNC system can meet the requirement of complex application.

6 Parameter

The difference between K2T and K1TBⅡ is described below.

0	**0**	**4**				**PUCH**			

PUCH 1 : RS232 is effective
Note1：This parameter require Power off.
Note2：If no RS232,this parameter must not be set.
Note3：The parameter SOVI of K1TB is rearranged into P042 .

0	**4**	**2**					**DPLS**		**SOVI**

DPLS 1：Select the double direction pulse in motion command.
0：Select the sigle direction pulse in motion command.
SOVI 1：Select external Feerate override switch, and the one from the panel is ineffective.

0	**5**	**7**	**BAUTE**

BAUTE: Set the baute rate of RS232

7 Diagnose status

There arethree diagnose status pages. Each page contains 16 diagnose status byte.

7.1 INPUT

0	0	0			*DECX	X16				
0	0	1			*DECZ	*ESP1	T04	T03	T02	T01
0	0	2					*ESP2	*SPL	*SPK	ST
0	0	3	X37	X36	X35	X34	T08	T07	T06	T05
0	0	4	X47	X46	X45	X44	X43	X42	X41	X40
0	0	5	X57	X56	X55	X54	X53	X52	X51	X50
0	0	6	X67	X66	X65	X64	X63	X62	X61	X60
0	0	7	X77	X76	X75	X74	X73	X72	X71	X70
0	0	8	X87	X86	X85	X84	X83	X82	X81	X80
0	0	9	X97	X96	X95	X94	X93	X92	X91	X90
0	1	0	X107	X106	X105	X104	X103	X102	X101	X100
0	1	1	X117	X116	X115	X114				

7.2 OUTPUT

0	1	6	SPZD	TL－	TL＋	M32	M08	Y12	M04	M03
0	1	7	QPJ	QPS	Y25	ESPO	S4	S3	S2	S1
0	1	8	Y37	Y36	Y35	Y34	Y33	Y32	Y31	Y30
0	1	9	Y47	Y46	Y45	Y44	Y43	Y42	Y41	Y40
0	2	0	Y57	Y56	Y55	Y54	Y53	Y52	Y51	Y50
0	2	1	Y67	Y66	Y65	Y64	Y63	Y62	Y61	Y60
0	2	2	Y77	Y76	Y75	Y74	Y73	Y72	Y71	Y70

7.3 MDI key status

			7	6	5	4	3	2	1	0
0	2	4								
0	2	5	*SPK	RST	W	P	T	M	9	8
0	2	6		CAN	EOB	SHT		DEL	INS	ALT
0	2	7					PGU	PGD	CRU	CRD
0	2	8	TOL	COL		SPL	F0		MOD	DSP
0	2	9				RT	+Z	−Z	+X	−X

7.4 System Status

0	3	2	HX/RV1		*DECX		−X	+X		
0	3	3	HZ/RV2		*DECZ		−Z	+Z		
0	3	4	DRN				GR2	GR1		
0	3	5	MLK	MP2	MP1		SBK	BDT		
0	3	6	ZRN	*SSTP	SOR	SAR	FIN	ST	STLK	MIR1
0	3	7	ERS	RT	*SP	*ESP	*OV8	*OV4	*OV2	*OV1
0	3	8	PN8	PN4	PN2	PN1	KEY	MD4	MD2	MD1
0	3	9	CDZ	SMZ	AFL	OVC		SOVC	SOVB	SOVA
0	4	0			RFZ	RFX		PCS	PCZ	PCX
0	4	1	FPA3	FPA2	FPA1	FPA0	EXB2	EXB1	ALMZ	ALMX

FPA3－0：The hardware version

EXB2、1：The mark of additional DI/DO

0	4	2	Analog spindle output

0	4	4		CSCT	CITL	COVZ	CINP	CDWL	CMTN	CFIN

0	4	5	STP	REST	EMS		RSTB			CSU

0	4	6	RERRX

0	4	7	RERRZ

RERRX the accumulated error axis or pulse of current step along X

RERRZ the accumulated error axis or pulse of current step along Z

Ⅶ K1TBⅢCNC system

The programing , operation and function in K1TBⅢ CNC system is the same as in K1TBⅡ CNC system. However the dimension is different . In this section , the difference is specified.

1 K1TBⅢ Installation Dimension of CNC Control unit

1.1 Panel

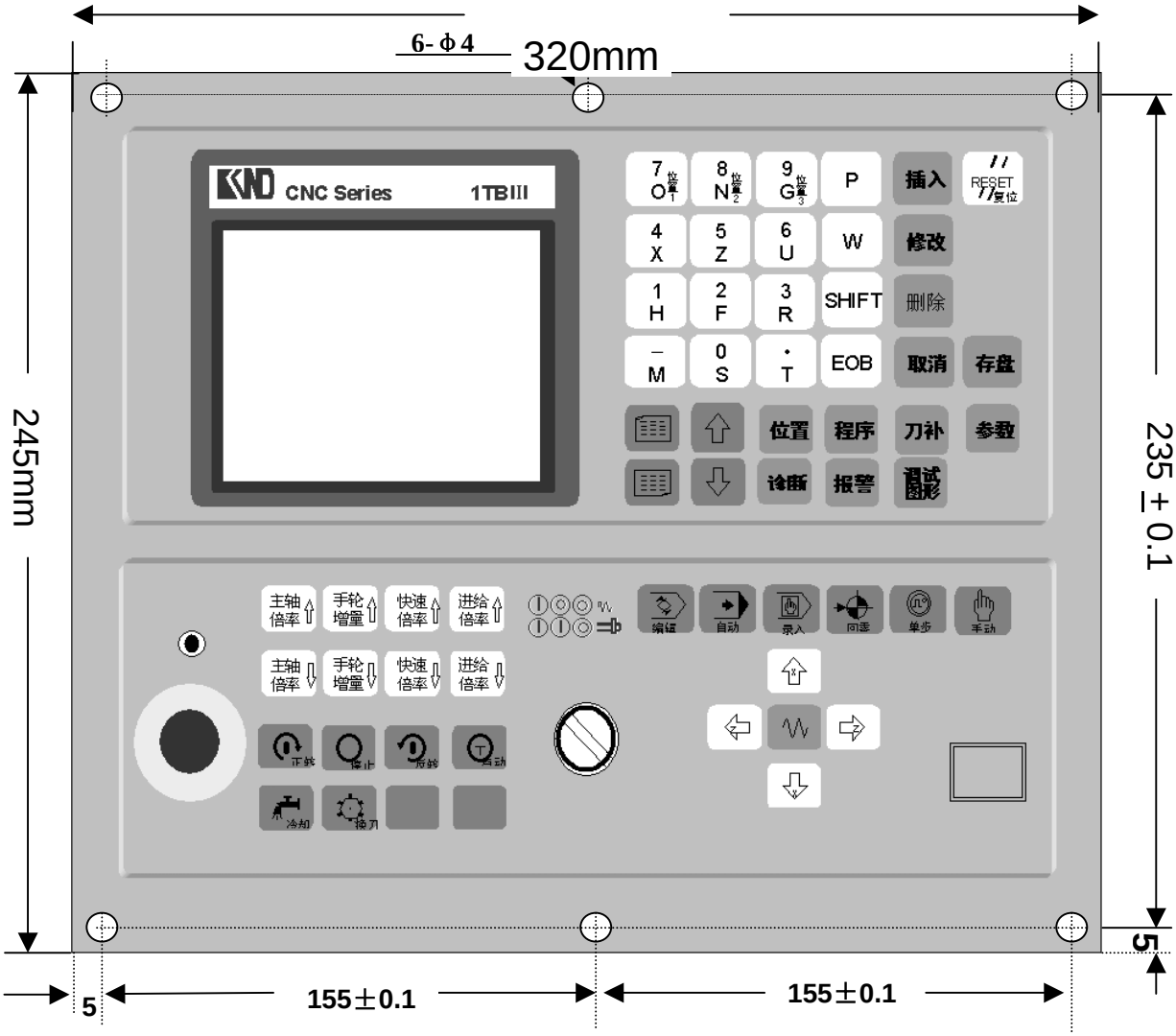


Fig. 1. 1 Frong view

1.2 Upside view of K1TBⅢCNC system

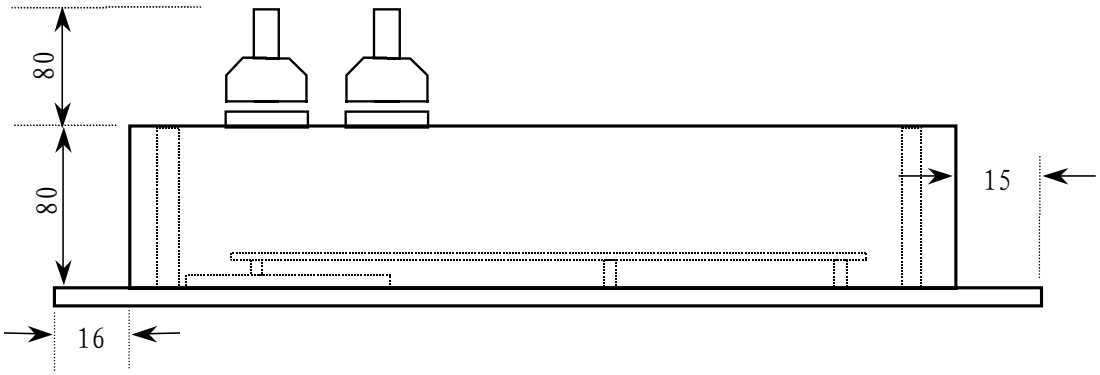


Fig. 1.2 Upside view

1.3 Left view of K1TBⅢCNC system

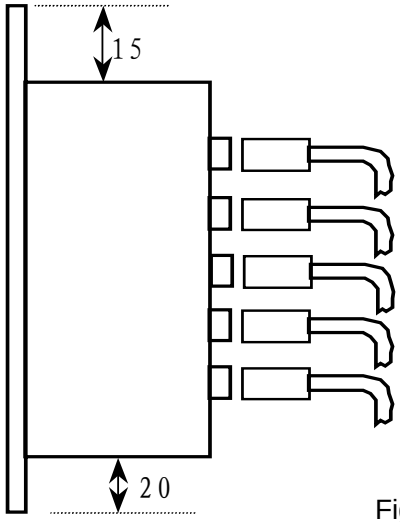


Fig. 1.3 Left view

2 Back cover Diagram