This user manual describes all proceedings concerning the operations of this CNC system in detail as much as possible. However, it is impractical to give particular descriptions for all unnecessary or unallowable system operations due to the manual text limit, product specific applications and other causes. Therefore, the proceedings not indicated herein should be considered impractical or unallowable.

PREFACE

Your Excellency,

It's our pleasure for your patronage and purchase of this GSK983Ta CNC system made by GSK CNC Equipment Co., Ltd.

This book is divided into Text and Appendix. The text contents include the specification and programming and operation of the system, and the appendix contains G codes, parameters, alarm messages and so on.

Safety Warning

Accident may occur by improper connection and operation!

This system can only be operated by authorized and qualified personnel. Please carefully read this manual before usage!

Special caution:

The power supply fixed on/in the cabinet is exclusively used for the CNC system made by GSK.

It can't be applied to other purposes, or else it may cause serious danger!

ANNOUNCEMENT

■Delivery and storage

- Do not put over six packing boxes in piles
- Never climb the packing box, stand on it, or place heavy items on it
- Do not use cable connected with the product to drag or carry products
- Take particular care of the front panel and the display of the system
- Moistureproof, exposure and rain measures are needed

■Check before acceptance

- Confirm the required products after opening the package
- Confirm the products are not damaged in transportation
- Confirm all parts are full and not damaged in accordance with the detailed list
- Please contact with GSK when the product type is inconsistent with the required, or the accessories are lacked or the products are damaged in transportation

■Wiring

- Only qualified persons can connect the system or check the connection
- The system must be earthed, and the earth resistance must be less than 0.1Ω. The earth wire cannot be replaced by zero wire
- The connection must be correct and firm to avoid any fault or unexpected consequence
- Connect with surge diode in the specified direction to avoid damage to the system
- Switch off power supply before plugging out or opening electric cabinet

■Troubleshooting

- Switch off power supply before troubleshooting or changing components.
- Check the fault when short circuit or overload occurs. Restart can only be done after troubleshooting.
- Frequent switching on/off of the power is forbidden, and the interval time should be at least 1 min.

ANNOUNCEMENT!

- **This manual describes various possibilities as much as possible. However, operations allowable or unallowable cannot be explained one by one due to so many possibilities that may involve with, so the contents that are not specially stated in this manual shall be regarded as unallowable.**

WARNING！

- **Please read this manual and a manual from machine tool builder carefully before installation, programming and operation, and strictly observe the requirements. Otherwise, products and machine may be damaged, workpiece be scrapped or the user be injured.**

CAUTION！

- **Functions, technical indexes (such as precision and speed) described in this user manual are only for this system. Actual function configuration and technical performance of a machine tool with this CNC system are determined by machine tool builder's design, so functions and technical indexes are subject to the user manual from machine tool builder.**
- **Though this system adopts standard operation panel, the functions of the keys on the panel are defined by PLC program (ladder diagram). It should be noted that the keys functions described herein are for the standard PLC program (ladder diagram).**
- **For functions and effects of keys on control panel, please refer to the user manual from machine tool builder.**

This manual is subject to change without further notice.

Safety Responsibility

Manufacturer's Responsibility

——Be responsible for the danger which should be eliminated and/or controlled on design and configuration of the provided CNC systems and accessories.

——Be responsible for the safety of the provided CNC systems and accessories.

——Be responsible for the provided information and advice for the users.

User's Responsibility

——Be trained with the safety operation of CNC system and familiar with the safety operation procedures.

——Be responsible for the dangers caused by adding, changing or altering on original CNC systems and the accessories.

——Be responsible for the failure to observe the provisions for operation, adjustment, maintenance, installation and storage in the JOG.

This manual is reserved by end user.

This manual is reserved by end user.
We are full of heartfelt gratitude to you for supporting us in the use of GSK's products.

Contents

Chapter 1 Summary

1.1 Summary

GSK983Ta Turning CNC System (hereafter referred to as **"CNC System").** It is a new CNC controller developed by GSK CNC Equipment Co., LTD based on market and user's requirements. Characterized with high accuracy, high capability, fixed-software, two axes linkage, close loop (semi-close loop or close loop), embedded PLC function, it is widely applied to CNC Lathe, CNC wire cutting machine tool and CNC Cylindrical grinding machine tool etc.

GSK983Ta Turning CNC system is employed with high speed MPU in control circuit, exclusive LSI, semiconductor memory and newly storage parts, improving greatly the reliability, and the performance-price ratio.

The system can be matched with AC servo motors widely applied in the world, using a high-capacity pulse encoder as a detecting element, which forms a close loop CNC system.

In the manual, the **GSK983Ta** Turning CNC system's programming, operation and parameter explanation are described.

System selection functions are also described at one time in this manual, but all selection functions are not always included in the actual equipment. Refer to the manual issued by the machine tool builder when the user uses the functions.

1.2 Precautions

CNC machine tool control function, is not only determined by the system function of CNC controller, also by the machine tool's strong circuit, servo equipment, CNC controller and machine operation control. The manual cannot narrate the combination of control functions, programming and operations in detail, it explains the CNC system function, and control functions about all kinds of machining lathe in detail. The user must refer to the manual issued by the machine tool builder.

The proceedings narrated in this manual are prior to the manual issued by machine tool builder.

In the manual, we have tried as much as possible to describe all the proceedings; however we cannot describe all matters, otherwise, it makes the manual more complex. So, some often used functions are explained correspondingly in the manual.

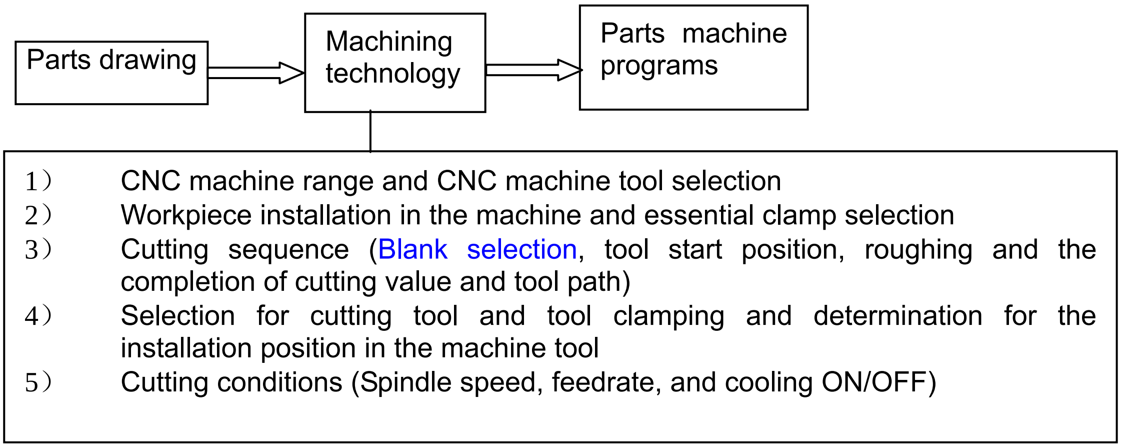
In the part of notes of this manual, some items are especially explained. You can skip some notes without special explanations in the manual to read the followings, and at last read it until you have read the followings.

Chapter 2　Programming

2.1　What is Programming

CNC machine tool is to control the machine and perform operations according to the compiled program beforehand. Tool path and other machine conditions should be programmed before parts are machined by the CNC machine tool, and the compiled program is called **Parts Program**.

The process from part drawing to part machining is shown below:



Process table

N	G	X	Y	…	…	…
N001	G00	X800	Y200			
N002		X50	Y20			

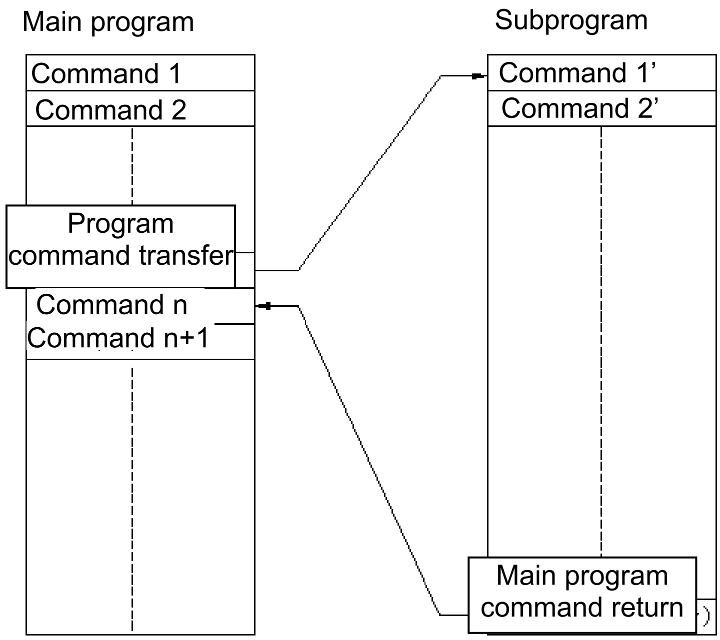
Compile contents of the process table into programs to into the CNC.

The followings describes a part program to be compiled.

2.2　Structure of an NC Program

A program includes a main program and subprograms. Normally, the CNC operates according to the main program. However, when a command edited by the main program calls a subprogram in the main program, then the CNC performs operations according to the subprogram. Besides, when the subprogram is commanded to return to the main program, the CNC performs operations based on the main program's commands.

The CNC memory can store 191 main programs and subprograms and the CNC system operates one of them to run the machine tool.



Note : As for storage and selection modes of programs, refer to the “CHAPTER FOUR OPERATION”.

2.2.1 Block

A program is composed of multiple commands, and one group of commands is called a block. Two blocks are separated by the end code of block, and the block end code is represented by “;” character in the following explanation of the manual:

For example:

N2 G00 X10.0 Z22.0；

N3 G01 Z10.0 R5.0 F100；

N4 X38.0 K-4.0；

N5 Z0；

Note 1: Character number of a block is not restricted.

Note 2: Block end code: EIA code is used with CR, ISO code is used with LF.

2.2.2 Word

A block consists of words. A word is composed of an address and its following several digits. (+/- may be prefixed to a digit).

$$\underbrace{\underbrace{X}_{\text{Address}} \quad — \quad \underbrace{1000}_{\text{Number}}}_{\text{Word}}$$

An address uses one of letters (from A to Z) to defines meanings of its following digits. The following table describes addresses and meanings used in the CNC. One address’ meaning is

different because preparatory functions have different commands.

Function	Address	Meaning
Program No.	O —— EIA ： —— ISO	Program number
Sequence No.	N	Sequence number
Preparatory function	G	Specifies a motion mode (linear, arc ect.)
Coordinate word	X，Z，U，W	Coordinate axis movement command
	R	Arc radius, corner R
	I，K	Coordinate of arc center, chamfering
Feedrate	F，E	Feedrate command, leading
Spindle function	S	Spindle speed command
Tool function	T	Tool number, tool offset number
Miscellaneous function	M	Switch ON/OFF control command on machine side
Dwell	P，X，U	Dwell time
Program no. designation	P	Subprogram number
Number of Repetitions	L	Number of subprogram repetitions, number of canned cycle repetitions
Parameter	A，D，I，K	Canned cycle parameter
Sequence no. designation	P，Q	Sequence number for specifying cycle program start and end

The following block can be formed by using these program words:

N___G___X___Z___F___S___T___M ；

N	G	X Z	F	S	T	M	；
Sequence no.	Preparatory function.	Coordinate word	Feedrate	Spindle function	Tool function	Miscellaneous function.	Block end code

In the following example for program table, one row indicates as a block, one grid of a block indicates as a program word.

<table>
<tr><td colspan="3">Name Test program 1</td><td colspan="8" rowspan="2">Program record</td><td colspan="4">Date:</td><td>Page1/1</td></tr>
<tr><td colspan="3">Program no. 0(；) 4321</td><td colspan="4">Programmer:</td><td>Confirm ；</td></tr>
<tr><td>/</td><td>N</td><td>G</td><td>X</td><td>Z</td><td>U</td><td>W</td><td>R/I</td><td>K/A/D</td><td>F/E</td><td>S</td><td>T</td><td>M</td><td>P</td><td>Q</td><td>L</td><td>；</td></tr>
</table>

Chapter 2 Programming

	N10	G50	X220.0	Z190.0												;
	N11	G00	X176.0	Z132.0												;
	N12	G72			U4.0	W2.0		D700	F30	S400			P13	Q18		;
	N13	G00		Z58.0						S600						;
	N14	G01	X120.0			W12.0			F15							;
	N15					W10.0										;
	N16		X80.0			W10.0										;
	N17					W20.0										;
	N18		X86.0			W22.0										;
	N19	G70											P13	Q18		;
	N20											M02				;

2.2.3 Format Input

Each program word composed into a block should be in accorded with the following format. If the input is variable block format, the number of program word in the block and word number in a programming word can be changed. It is very convenient in programming, and GSK983Ta CNC system uses this format.

1） Metric input

N04· G02·αL+ 053· βL+053· { RD053 · / ID053KD+ 053 } { F032 · E034· / F050 } · { S02 / S04 / ? } ·M02 { T03 / T04 }

2）Inch input

N04· G02·αL+044· βL+044 · { RD044 · / ID044·KD044 } { F024 ·E016· / F032 } { S02 · / S04 } { T03 · M02 / T04 }

Note 1: α：X or U;

β：Z or W

Note 2: Address and digit meanings are as shown below:

X L + 0 5 3

- 3 — The 3 digits in the following of decimal point
- 5 — The 5 digits in the front of decimal point
- 0 — Leading zero can be omitted
- + — With sign
- L — Absolute or incremental value
- X — Address

I　D　0　5　3

- 3: The 3 digits in the following of decimal point
- 5: The 5 digits in the front of decimal point
- 0: Leading zero can be omitted
- D: Incremental value with sign
- I: Address

Example：

To move the X axis to 50.123mm at the rapid traverse, the tool move commands are shown below:

G00　X　50　123

- 123: 3 digits in the following of the decimal point
- 50: The 5 digits in the front of decimal point, the leading zero of 00050 can be omitted.
- G00: Even the leading zero can be omitted, G00 can be omitted to G0 (G00 specifies rapid feedrate)

Note 3: In one block, same address word commands more than two, in fact, the following commands are effective, no alarm.

For example: G01　M03　S200　M08；

M08 enabled, M03 disenabled.

1) G code in each group is also enabled at last.

2) R, I and K are simultaneously commanded in arc command, it is regardless of sequence, and the R is enabled.

3) E and F are commanded in thread cutting in a same block, the posterior commands are enabled.

4) X or U and Z or W, the posterior one commanded is enabled.

Note 4: F032 and F050 input by metric are shifted with parameter, also F033 and F051, refer to (2.4.4 feedrate 1/10) F032 (input by mm) and F024 (input by inch) are feed input format per rev. F050 (input by mm), F032 (input by inch) are feed input format per min.

Note 5: A, P, Q, L and D have many significations, so the above formats are omitted.

Note 6: Refer to (2.2.4 decimal point input) when inputting by using decimal point.

Note 7: The input values with parameter setting X, Z, O, W, I, K, R and D millimeter can be used with 10 multiple units.

$$\mathbf{\alpha L+\ 052\cdot\beta L+052\cdot}\left\{\begin{matrix}\mathbf{RD052}\\ \\ \mathbf{ID052\cdot\ KD052}\end{matrix}\right\}\quad\text{（Input in Millimeter）}$$

Refer to (Section 2.3.2.3, 10 multiple of input unit).

Note 8: The standard configuration function is S4 bit numeric analog spindle function.

2.2.4　Decimal Point Programming

Numerical values can be entered with a decimal point in this device. A decimal point can be used when entering a distance, time, or speed. But all addresses have limitation; the position of decimal point is indicated as the positions of mm, inch, deg and s (second).

Z15.0　　Z15mm or Z15inch

F10.0　　10mm/r, 10mm/min or 10inch/min, 10inch/r

Chapter 2　Programming

G04 X1. Dwell 1 second

Decimal points can be specified with the following addresses: X, Z, U, W, I, K, R, E and F.

Note 1: X and U can be input by decimal point, but P cannot (because the P can be employed for sequence number), when commands dwells

Note 2: The corresponding G code should be commanded prior to the number within one block.

1) G20; (Inch specifying)

X1.0 G04; X1.0 G04 is equivalent to is not represented to t X10000 G04 when instruct in inch, the result dwell time is 10s.

G04 X1.0; It is regarded as G04X1000, dwell time is 1s.

2) G98; (mm/min specifying)

F1.G99; Specifying F1 G99 into 0.01mm/rev. (G99 is mm/rev.)

G98: (mm/min specifying)

G99 F1 Specifying G99F100 into 1mm/rev. (G99 is mm/rev.)

Note 3: It is very different with or without decimal point. Note that it is also different to the computer programming.

G21; (Specifying mm)

X1.……X1mm

X1……X0.001mm

G20; (Specifying inch)

X1.……X1inch

X1……X0.0001inch

Note 4: The numbers with or without decimal point can be mixed to use.

X1000 Z23.7;

X10. Z22359;

Note 5: The values following the least setting unit is specified, the values less than the least setting unit is omitted. When specifying X1.23456 when the metric input is regarded as X1.234, the inch input is regarded as 1.2345. There is cumulation error when the incremental command occurs. There no cumulation error but has omittance error when the absolute command occurs. Same, digit can not over the max. allowance digit.

X1.23456789……it overs 8 digits, alarm occurs; X1.2345678…… it is within 8 digits, the alarm will not generated, but the value less than the min. setting unit will be striped away.

Note 6: When a number with decimal point is input, A minimum incremental integer is shifted by this number.

For example: X12.34→X12340 (mm input)

The shifted integer value should be verified with digit.

2.2.5 Max. Command Value

The max. command values for each address are shown below, but, these figures represent limits on the CNC side, which are totally different from limits on the machine tool side. For example: for the CNC equipment, the movement value of X axis is 100m (in the occasion of metric), in fact, an actual stroke along the X axis may be limited within 2m, the feedrate is not changed. The CNC cutting feedrate can be controlled within 30m/min, and the machine tool actually can be limited within 6m/min. Refer to the manual and machine tool manual in actual programming. Programmer can perform programming after fully understanding program limitations.

Table 2.5 Basis address and ranges of command values (including additional option)

Function		Address	Input in mm Output in mm	Input in inch Output in mm	Input in mm Output in inch	Input in inch Output in inch
Program No.		:(IS0) O (EIA)	1～9999	Same as left	Same as left	Same as left
Sequence No.		N	1～9999	Same as left	Same as left	Same as left
Preparatory function		G	0～99	Same as left	Same as left	Same as left
Coordinate word		X, Z U, W I, K	±99999.999 mm	±3937.0078inches	±99999.999mm	±9999.9999inches
		R	0~99999.999 mm	0~3937.0078inches	0~99999.999 mm	0~9999.9999inches
Feed per revolution Thread leading		F	(1) mm/r ～500.00mm/r	0.0001inches/r ～50.0000inch/r	0.01 mm/r ～ 500.00 mm/r	0.0001 inches/r ～ 50.0000inches/r
Feed per revolution, thread leading (feedrate 1/10) (Parameter setting)		F	0.001mm/r ～ 500.000mm/r	Ditto	0.001 mm/ r～ 500.000mm/ r	Ditto
Feed per minute		F	1 mm/min ～ 15000mm/min	0.01 inches/min ～ 600.00inches/min	1 mm/min ～ 15000 mm/min	0.01 inches/min ～ 600.00inches/min
Feed per minute (feedrate 1/10) (Parameter setting)		F	0.1 mm/min ～ 15000.0mm/min	Ditto	0.1 mm/min ～ 15000.0mm/min	Ditto
Thread leading		E	0.001 mm ～ 500.0000mm	0.000001 inches～ 9.999999inches	0.0001 mm～ 500.0000mm	0.000001 inches～ 9.999999inches
Thread leading (feedrate 1/10) (parameter setting)		E	0.00001～ 99.99999mm	Ditto	0.000001～ 99.99999mm	Ditto
Spindle function		S	0～9999	Same as left	Same as left	Same as left
Tool function		T	0～9932	Same as left	Same as left	Same as left
Miscellaneous function		M	0～99	Same as left	Same as left	Same as left
Dwell		X, U, P	0～99999.999S	Same as left	Same as left	Same as left
Designation of a sequence No.		P, Q	1～9999	Same as left	Same as left	Same as left
Number of repetitions		L	1～9999	Same as left	Same as left	Same as left
Angle		A	Specifying value	Same as left	Same as left	Same as left
Parameter	Cutting value	D, I, K	Some coordinate word	Same as left	Same as left	Same as left
	Times	D	1～9999	Same as left	Same as left	Same as left

Note: Feedrate per rev. and thread leading are actually related to spindle speed. F/min. is determined by the converted speed, and is specified by above-mentioned command. Parameter of Feedrate 1/10 is the 3rd bit of NC No.5.

2.2.6 Program Number

The CNC control can store many programs into the NC memory, program numbers are added to distinguish from these programs.

Program number definition is shown as below:

O $\underbrace{\square\ \square\ \square\ \square}_{\text{4 digits}}$ (Program number can be from 1 to 9999)

Program performs with a program number, till to M02;, M30; or M99; end.

Chapter 2 Programming

O1111………………………………M02;

$\underbrace{\hspace{6cm}}$

#1111 program

O2222………………………………M30;

$\underbrace{\hspace{6cm}}$

#2222 program

M02 or M30 indicate that the main program ends; M99 indicate that subprogram ends.

O5555…………………………………M99;

$\underbrace{\hspace{6cm}}$

#5555 subprogram

Note 1: ":" is replaced "O" when ISO code is performed.
Note 2: At the end of the program, the blocks with optional block skip codes such as /M02;, /M30;, M99 are disabled .
Note 3: There is no any program number at the beginning of a program, the sequence number of the program beginning is regarded as the program number; but the NO replaces the program number is unallowed.
Note 4: When neither program number nor sequence number is at the beginning of the program, the program number should be specified when storing to the memory. (Refer to Chapter 4, Section 4.5.15).
Note 5: The subprogram should have program number.
Note 6: Program numbers 9000~9899 are employed by machine tool builder, the user can not use.
Note 7: When selecting with mechanic, the programs 9900~9999 are employed with the data of mechanic.
Note 8：In the end of the program, if there is not M02, M30, M99 command, but with ER（EIA）or %（ISO）and the next block number, the program end is set by No. 306 bit3（NEOP）.

2.2.7 Sequence Number

At the head of a block, a sequence number is specified by a number (1~ 9999) following the address N with less than 4 digits. Sequence of sequence number can be continuous or not. All blocks or some blocks with necessity have sequence numbers.

It is suggested that the sequence number should be specified at key position. For example, the tool change and new tool are performed.

Note 1: In order to be compatible with other NC devices, the sequence number N0 should not be employed.
Note 2: 0 is disabled as program number for some devices, so 0 can not be used for sequence number of program number.

2.2.8 Optional Block Skip

When a slash followed by a number /n (n=1~9) is specified at the head of block and corresponding block skip BLOCK SKIP2 switch (optional skip block) is set to 1, the block with /n is ignored.

When optional skip block switch is set to 0, the block with /n is valid, namely, the operator can perform the block skip for the block with /n.

1 in /1 can be omitted. Press "Skip" key on the operation panel to determine whether the block is ignored.

When "Optional skip block 2=1", the omitted area is shown below:

；/2N123G01X4………………；N7856

Omittance area

Example：

N100 X100；

N101/2 Z100；

N102/2/3 X200；

N103/3 Z200；

In the above examples, blocks N101 and N102 are skipped when “Optional block skip 2=1”, and blocks N102 and N103 are skipped when “Optional block skip 3=1”.

Note 1: A slash (/) must be specified at the head of a block, if a slash is placed elsewhere in the block, the information from the slash to EOB code is ignored, however, the information in front of the slash is still enabled.

Note 2: When an optional block skip switch is on, TH and TV checks are made for the skipped portion in the same way as when the optional block skip switch is off.

Note 3: The optional block skip is processed when it is transferred from memory to cushion memory. When the slash / is specified at the head of a block is read into a cushion, even if the block skip switch is ON, this block will not be ignored.

Note 4: This function is still enabled during the sequence number searches.

Note 5: This function is disabled when the program is stored, regardless of how block skip switch is set.

Note 6: When the program in the memory is output to the outer PC by communication interface, regardless of how block skip switch is set, it can be output integrally.

Note 7: Some optional block skip switch may not be used for some machine tools. So please refer to the machine tool builder manual.

Note 8: When an optional block skip is specified, if multiple optional block skip codes (/) are specified in a block, 1 in /1 can not be omitted.

Please describe /1 according to the above.

Example：

Error：/ /3G00 X10.0；

Right：/1/3G00 X10.0；

2.3 Dimension Word

The dimension word describes tool movement, composed of movement axis address, its value indicates move direction and length. The value modification is determined by the absolute and incremental programming. (Refer to Section 2.3.7)

Dimension word address		**Meaning**
Basic axis	X, Z	Describing a target position in coordinate system (Absolute programming).
	U, W	Specifying a move distance (incremental programming). U indicates X axis, W indicates Z axis.
Parameter for	R	Specifying an arc radius.

arc interpolation	I, K	Specifying a distance (I) or (K) from starting point to arc center along the X axis or Z axis

For example: U-200.0 tool moves to 200.00 along X axis negative direction from current position.

2.3.1 Controlled Axes

Movement axes of machine tool controlled by CNC system are called Controlled axes. Each controlled axis is called by the controlling dimension word. There are two controlled axes X and Z.

There are 2 simultaneous controlled axes are specified in a block.

Controlled axes	Simultaneous controlled axes
X, Z	2 pieces

2.3.1.1 Coordinate axis and movement sign

If a machine tool offers multiple relations between machine tool coordinate axis and tool movement sign, severe mess may occurs in a programming. In order to simplify the programming and guarantee the versatility of the program, the unified criterion (EIARS-267-A&ISO841) is applied for the coordinate system of CNC machine tool and direction name, and the linear feed coordinate axes are indicated by X, Y and Z, they are normally called coordinate axes. The relationships among X, Y and Z are determined by the right hand rule, see Fig.3.1-1:

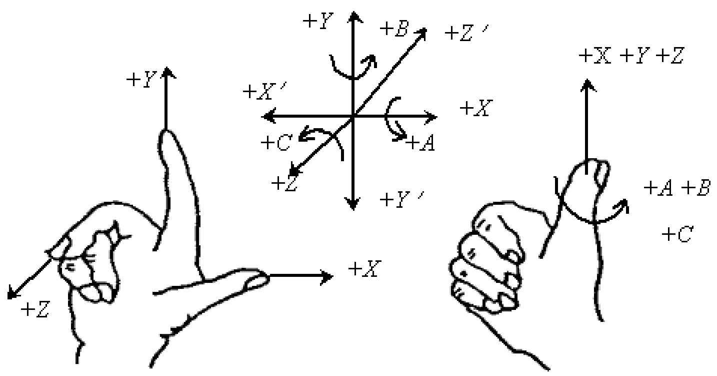


Fig.3.1-1 Machine tool coordinate axis and direction

A thumb direction is positive direction of X axis, a forefinger is positive direction of Y and a middle finger is positive direction of Z.

Circle feed coordinate axes rotated around with the X, Y and Z axes are separately indicated with A, B and C in terms of the right hand rotation rule, see the figure. The thumb direction is +X, +Y and +Z, and the direction of forefinger and middle finger are the circle feed movement +A, +B and +C directions.

Feed movement of CNC machine tool, some are operated by the spindle drives tool, and some are operated by the workpiece on the worktable. The abovementioned + coordinate axes are supposed to the workpiece stops, the tool is feed movement direction which is related to the

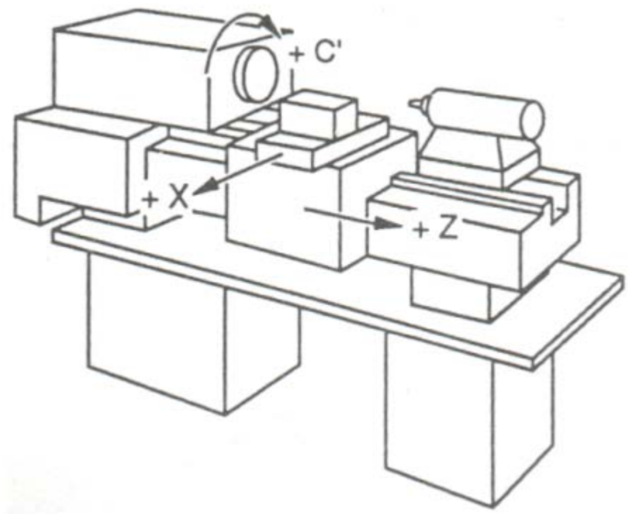
Chapter 2 Programming

workpiece. If the workpiece movement is specified, it is indicated with character“”In terms of relation of movement, the + workpiece movement direction is reversed to the + tool movement, that is:

+X =-X′, +Y =-Y′, +Z =-Z′,

+A =-A′, +B =-B′, +C =-C′

In a similar way, negative directions of both movements are reversed each other.



Direction on the machine tool coordinate axis is determined by the type and component layout of each part of the machine tool. For the turning machine:

—— Z axis is identical with the spindle axial line, the distance between parts and tool is enlarged by movement along + Z axis direction;

—— X axis is vertical to Z axis, which is corresponding to revolving tool post movement, the distance between parts and tool is enlarged by movement along + X axis direction;

—— Y axis (it is nominal) consists of right hand coordinate system rule with X axis and Z axis.

Note the following items during programming.

a) Program must refer the standard coordinate system (Right hand square coordinate).

b) Supposing the workpiece is not move during programming, the tool moves around the tool.

2.3.2 Increment system

Increment system is determined by the following two elements.

2.3.2.1 The least input increment (input unit)）

The minimum units are specified by the tool movement, these minimum units are specified by mm/inch.

2.3.2.2 Minimum movement unit (output unit)

Minimum movement unit of machine tool is specified, any group of the following can be employed by using mm, inch or degree units.

	The least input increment	The least movement unit
Input in mm, output in mm	0.001mm	0.001mm
Input in inch, output in mm	0.0001inch	0.001mm
Input in mm, output in inch	0.001mm	0.0001 inch
Input in inch, output in inch	0.0001inch	0.0001 inch

Specifying the diameter, the min. movement unit of X axis is also a diameter value.

Whether the least movement unit of machine tool is either 0.001mm or 0.0001inch is determined by the set parameter in advance (SCW) #5.0.

Whether the least input increment is either 0.001mm or 0.0001 inch is determined by the G code or set by MDI.

G20 The least input increment 0.0001 inch

G21 The least input increment 0.001mm

The mode of power on is still that of G20 and G21 before power on.

2.3.2.3 10-fold input unit magnification

The least input increment input in mm can be changed into 0.01mm by the parameter #5.1（MIC）. The least input increment input in inch is not changed.

	Address	Input unit	
		Input in mm	Input in inch
Dimension word	X，Z，U，W，R，I，K	0.01mm	0.0001 inch
Dwell time	X，P，U	0.01s	0.001s
Parameter	D	0.01mm	0.0001 inch

The following items are not be changed:

(a) Different input

(b) Display unit

(c) Range for max. command value

(d) Units for step and manual feed

(e) Offset input

(f) Others

Note 1: The input unit in the manual is either 0.0001 inch or 0.001mm.
Note 2: When input in metric is performed, the display unit is 0.01mm by setting the parameter #5.2 (MDL).

2.3.3 Maximum Stroke

The maximum stroke can be specified in the system, see the following table:

Input in mm Output in mm	Input in inch Output in mm	Input in mm Output in inch	Input in inch Output in inch
±99999.999mm	±3937.0078inch	±99999.999mm	±9999.9999 inch

Note: The above stroke should be changed with the different of machine tool, please refer to the machine tool manufacture's manual.

2.3.4 Program Origin and Coordinate System

The program origin and coordinate system should be confirmed in programming. Normally, the program origin is a random point at the workpiece.

For example, X axis origin is specified at the center of workpiece, and Z axis origin is specified at the left surface of workpiece terminal surface.

See Fig.3.4-1

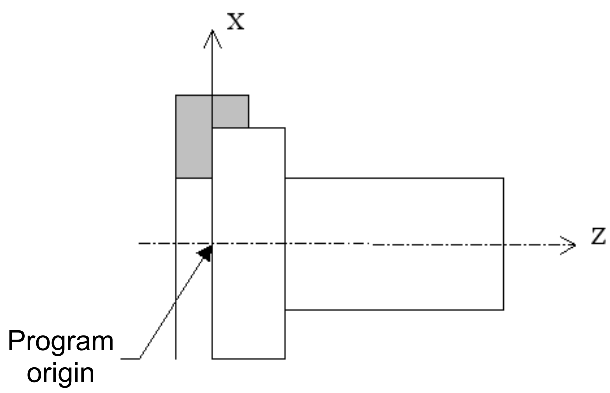
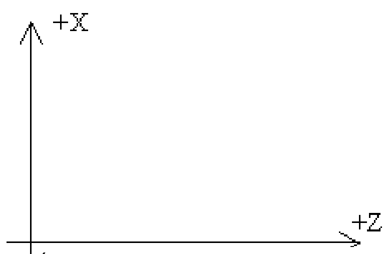


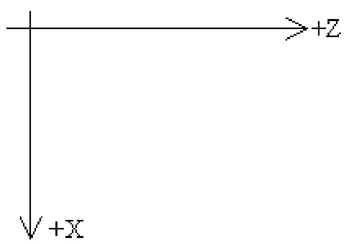
Fig. 3.4-1 Program origin

This coordinate system is called workpiece coordinate system.

There are two coordinate systems, one is left hand, and the other is right hand. Right hand coordinate system is employed in the manual.



(Right hand coordinate system)



(Left hand coordinate system)

2.3.5 Coordinate System and Machining Starting point

Workpiece coordinate system should be memorized to CNC during programming, the tool moves

and a program operates should be performed from a starting point. But, the starting point and tool coordinate value should be offered by using G50.

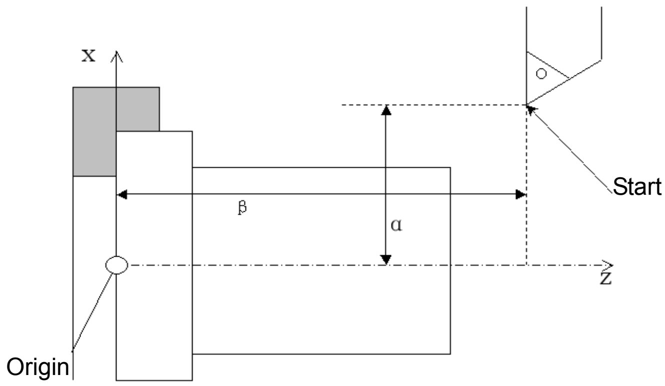


Fig. 3.4-2 G50 Workpiece coordinate system setting

Workpiece coordinate system of NC is specified by G50 X αZβ command.

Firstly, X=α and Z=β are applied to CNC, this command is offered at the head of a program or, the command is offered from MDI when the tool is at start position. (Refer to the CHAPTER FOUR, section 4.4.9 MDI run mode).

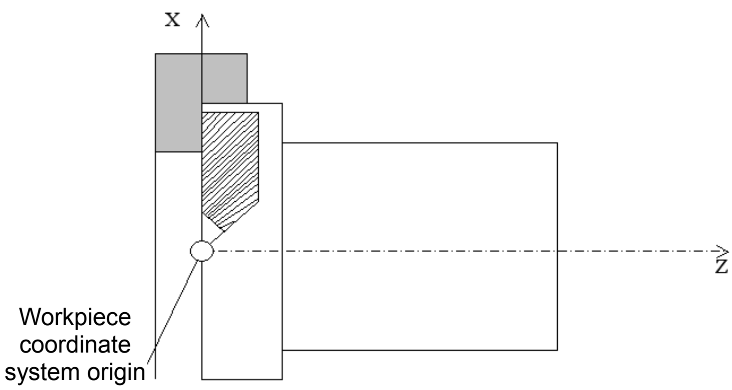
It is very easy to position to the starting point if the workpiece coordinate system is firstly confirmed:

(1) MDI/LCD display panel, command G00X αZβ; (G00 is positioning command).

(2) Manual feed moves the tool to X=α and Z=β from viewing position.

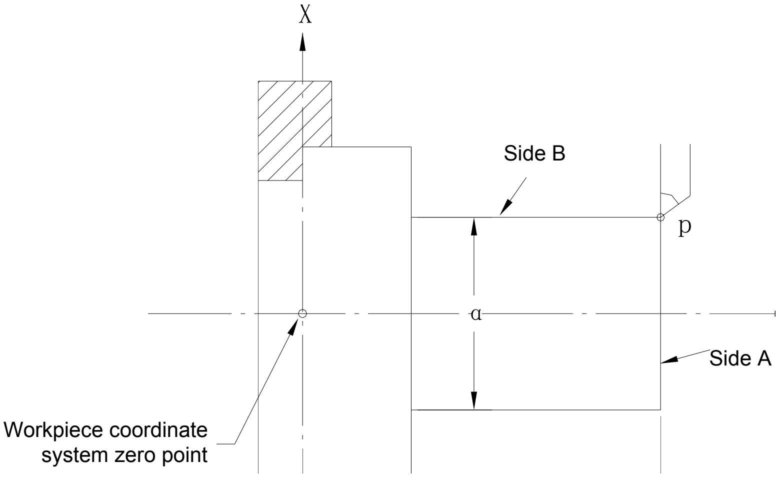
Workpiece coordinate system is determined by the following modes.

(Mode 1) Performing G50 in MDI mode.



When a tool is positioned at the origin of workpiece coordinate system, the following commands can be specified in MDI mode after positioning.

G50 X0 Z0；



Usually, it is impossible for the tool to position to the origin.

So, the tool is at the P position of workpiece in the abovementioned figure. Commands are performed in MDI mode below:

G50 XαZβ; the workpiece coordinate system is correspondingly set.

However, hard position of P point is the reason, so the following steps should be performed:

Manually cut Side A of workpiece.

The tool leaves and the spindle stops along the X axis, the Z axis does not move.

Measure the distance of “β” and specify G50 Zβ in MDI mode, press the DATA INPUT and CYCLE START keys to set the workpiece coordinate system into NC in MDI mode. The measure method of α value is similar to β.

(When side A and B are cut with a same tool, the offset value is 0; when different tools are employed, the difference between two tools is regarded as offset value to store. Offset value measure, see Section 2.6.5).

(Method 2) The coordinate system is set after positioning to the starting point is performed.

When the tool is positioned to the starting point, the following steps should be performed:

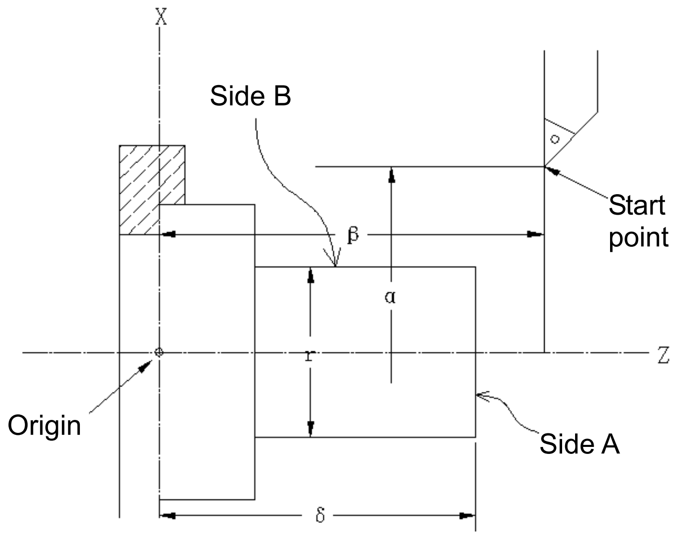
(1) Cut manually side A of workpiece.

(2) Z axis does not move, and the tool leaves from the X axis and the spindle rotation stops.

(3) Clear the relative coordinate value of Z axis (address W), (see Section 4.4.6).

(4) Measure the distance of “δ” in the following figure.

(5) Cut manually side B of workpiece.



(6) X axis does not moves, the tool leaves from the Z axis and the spindle rotation stops.

(7) Clear the relative coordinate value of X axis (Address U) (see Section 4.4.6).

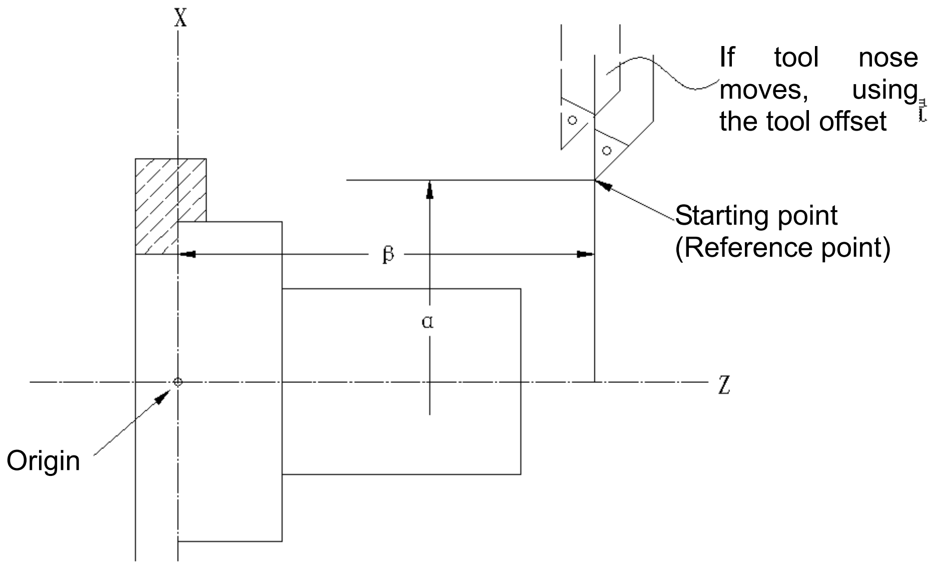
(8) Measure the distance of “r” above figure.

(9) View relative coordinate display and move the tool to X=α－r，Z=β－δcoordinate position at the same time.

α and β are coordinate value of starting point.

If G50 XαZβ is specified at the beginning of program, the coordinate system is set when the program is executed (in actual machining, offset setting is the same as that of method 1 when tool change is performed).

(Method 3) The reference point is regarded as the starting point at the head of program, and the coordinate system setting is performed.



When reference point returns, the tool starting point is coincident with reference point. The following steps are the same as of method 2, the tool nose can not be positioned at the reference point, but a standard tool can do so. Error of both can be set by tool offset value offset (see Section 2.6) or moving the workpiece coordinate system (see Section 2.5.10).

(Method 4) Automatic coordinate system setting.

When the corresponding parameter is set, a coordinate system is automatically set after the reference point return is performed.

Coordinate values α and β are separately set by parameter #375 and #376 when input in inch is performed（when input in inch，they are set by the parameter #379 and #380）.

The tool nose can not be positioned at the reference point, so, tool position can be compensated by using tool offset function (see the Section 2.6.1) or workpiece coordinate system movement function (see Section 2.5.10).

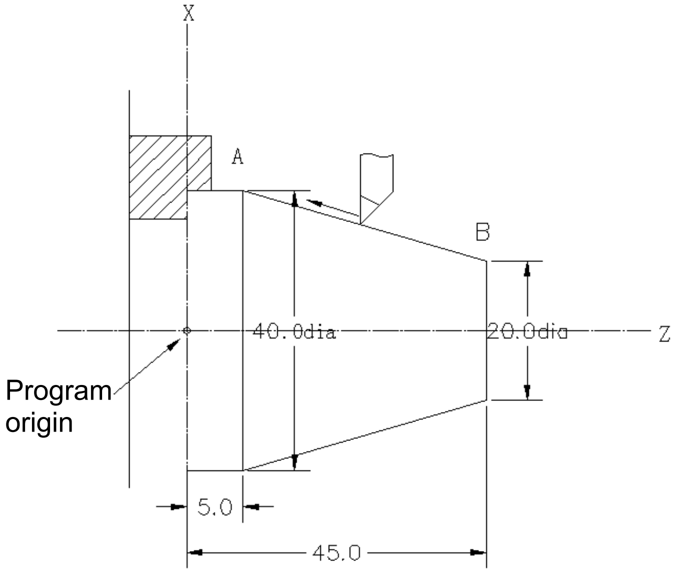
2.3.6 Reference Point

The reference point is a fixed position on the machine tool. The reference point return function is to perform the tool to return the reference point.

A program can not be randomly started from a certain position on workpiece coordinate system, but it can be started from the reference point accordingly. In this case, the reference point is a certain point on the machine tool, and the point corresponds to the zero of the workpiece coordinate system, and makes the program finish the machining normally. G50 command should be commanded when the tool returns reference point to set the workpiece coordinate system.

2.3.7 Absolute Command and Incremental Command

Command	**mode**	**Address**	**Command for B—>A in the following figure**
Absolute command	Specifying end point on workpiece coordinate system	X (coordinate value of X axis) Z (coordinate value of Z axis)	X40.0 Z5.0;
Incremental command	Specifying the distance from start to end points	U (distance of X axis) W (distance of Z axis)	U20.0 W-40.0

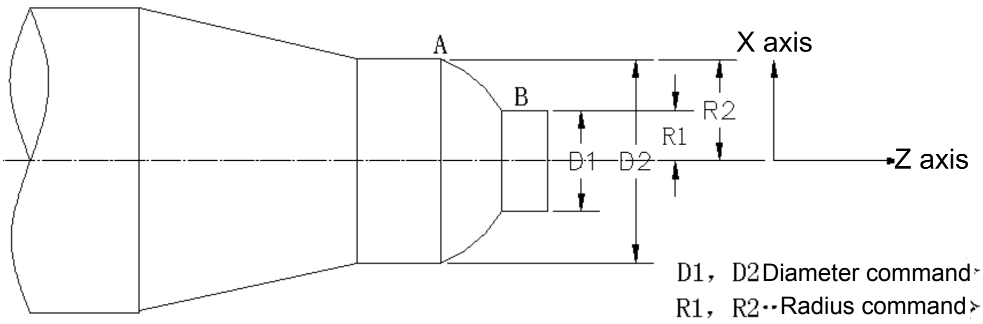


Note 1: Absolute command and incremental command are shared with the same block; the above example can be specified by the following command: X40.0 W-40.0;

Note 2: When X and U or W and Z are shared with the same block, the later specified is enabled.

2.3.8 Specifying Diameter and Radius

Usually, the cross-section of workpiece is round when NC turning machine is programming, so, two specifying dimension methods including diameter and radius values can be employed.



When the diameter is specified, it is referred to as diameter command; when the radius is specified, it is referred to as radius command. The radius program can be performed when the radius command is selected; when the diameter command is employed, some items should be noticed in the following table:

Item	Caution
Command of Z axis	Specifying diameter or radius irrelative
Command of X axis	Specifying with diameter
Incremental command by using address U	Specifying diameter value, from B to A, D1 to D2 in the above figure
Coordinate system setting (G50)	Specifying X axis coordinate value with diameter
X value of tool position offset	Setting diameter or radius value with parameter #8.7.

Parameter of cutting depth by using G90-G94 and G70-G76 along with X axis. (D, I and K)	Specifying radius value
Radius specifying for arc interpolation (R, I and K)	Specifying with radius
Feedrate along with X axis direction	Radius change/rev.; radius change/min.
X axis position	Diameter value display

Note 1: In the following explanation, command programming is not specified (diameter or radius). But, value of X axis is indicated with diameter value when the diameter command is performed. Value of X axis is indicated with radius value when the radius command is performed.

Note 2: When diameter value is used in tool offset and tool position offset value is changed to cut the outer diameter, the outer diameter is then changed with diameter value.

For example: Offset value changes for 10mm, and outer diameter changes 10mm at the diameter value, when tool keeps invariable.

Note 3: Tool position offset uses diameter value, the tool offset value can be set to the tool length value.

2.4 Feed Function

2.4.1 Rapid Traverse Rate

In rapid traverse mode, each axis on the machine tool moves at the specified rate.

Normally, the rapid traverse rate is confirmed and set by the machine tool builder before delivering the goods (set by parameter #92 RPDFX, #93 RPDFZ).

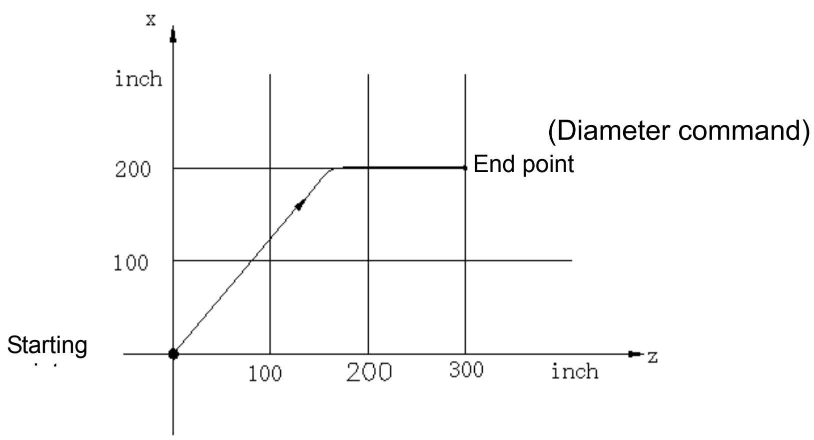
When each axis of machine tool separately runs, movement time of each axis from starting point to end point is different.

For example, when the rapid traverse rate of X and Z axes are 500inch/min and 800inch /min separately and their commands are shown below:

G00 U200.0 W300.0;

Movement time of X axis on the machine tool from the starting point is 12 seconds, Z axis is 22.5 seconds. If X axis value is specified with radius programming, the finished movement time is 24 seconds.

The tool path is shown below:

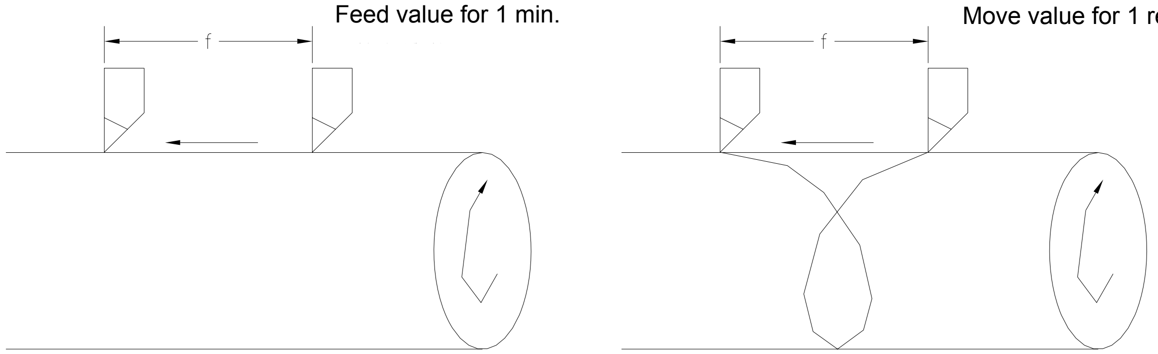


The switch on the machine operation panel can control override of rapid feedrate. (F0, 25%, 50% and 100%) F0 is set by parameter #13 RPDEL, and its unit is not indicated with (%), but indicated with mm/min. or inch/min.

2.4.2 Cutting feedrate

Feedrate per revolution or minute is determined by the digits followed by F.

Feed value for 1 min. Move value for 1 rev.



Feedrate/min.
(mm/min, inch/min)

Feedrate/rev.
(mm/rev, inch/rev)

		Feed/min.	Feed/rev.
Meaning		Tool feed distance/min.	Tool feed distance/rev.
Programming address		F	F
G code setting		G98	G99
Range	Input in Metric	1~15,000mm/min (F1~F15000)	0.01~500.00mm/r (F1~F50000)
	Input in Inch	0.01~600.00inch/min (F1~F60000)	0.0001~50.000 inch /r (F1~F500000)
Clamping value		Cutting feedrate is clamped at fixed speed. This value (Clamping value) is specified by the machine tool builder (Speed after the override is also clamped).	
Override		10%/step, modifying from 0~200%.	

The clamp value is specified by mm/min or inch/min. Cutting feedrate can be converted into mm/min or inch/min by the following formula when feed/rev is performed.

fm=fr×R

fm: Feed/min. (mm/min or inch/min)
fr: Feed/rev. (mm/r or inch/r)
R: Spindle speed (r/min)

Note 1: G98 and G99 are modal. When one of them is commanded, the command is always enabled before the other command is specified.

Note 2: The specified feedrate is set to the error within ±2% for NC computer, other than the acceleration/deceleration.

Note 3: Position encoder should be installed in spindle if the feedrate is feed/rev. when programming.

Note 4: F code input is allowed up to 7 digits. But the feedrate can not exceed the clamp value.
Note 5: If the number of revolution for position encoder is below 1, the cutting feedrate is uneven. So, the lower of the spindle speed is, the more uneven of cutting speed is.

2.4.3 Thread Lead

The digits following address F and E specifies a lead in thread cutting. Thread cutting is specified by G32, G76 or G92.

G code	Meaning
G32	Thread cutting
G92	Thread cutting cycle (Single canned cycle)
G76	Thread cutting cycle (Complex canned cycle)

Range of thread length, see the following:

Address	Input in Metric	Input in Inch
F	0.01 mm ~500.00mm	0.001 inch ~50.0000inch
E	0.0001 mm ~500.0000mm	0.000001 inch ~9.999999inch

Spindle speed limitation is as shown below:

$$R \le \frac{\text{Max.feedrate}}{\text{Thread lead length}} \quad (R \ge 1)$$

R: Spindle speed (r/min)

Lead: mm or inch

Max. feedrate: mm/min or inches/min

Max. feedrate is limited by least one of the following speed: (1) feederate per minute of Max. command speed; (2) Max. speed value is limited by motor or machine tool.

Note 1: The spindle speed is read from the position encoder installed on the spindle. Feed/min. is converted, and the machine tool moves with the converted speed.

Note 2: Speed override is disabled for the converted speed.
Note 3: The converted speed is clamped.

Address F specifies the thread lead, and the address F is the same as F used to specify feed/min. or feed/rev.

Address E is only used for specifying the thread lead.

The last one of thread leads specified by E and F is enabled.

When thread cutting is not processed, for feed/min and feed/rev, F code keeps enabled regardless of E code;

In thread cutting block, when E and F are in the same block, the latter is valid.

Example (program using the following sequence)	Enabled feedrate/ thread lead

G01Z…………….F100;	F100
G32Z…………….;	F100
G32Z…………….E200;	E200
G01Z…………….;	F100
Z…………….F300;	F300
G32Z……………;	F300
G32Z…………….F100E200;	E200
G32Z…………….E200F100;	F100

2.4.4 Feedrate 1/10

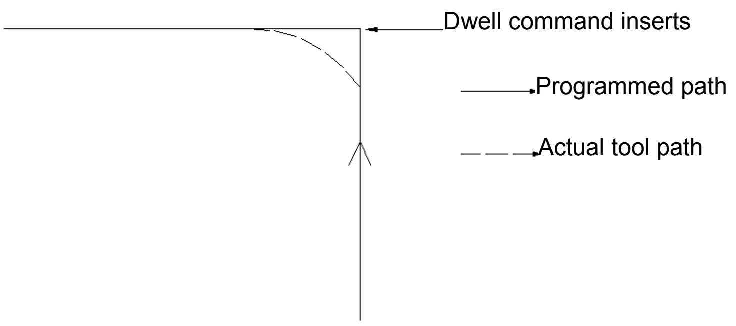
Feedrate input in mm can be shifted to the min. unit 1/10 by setting the parameter (FMIC).

Item	Min. input incremental	Range
Feed/min.	0.1mm/min	F1~F150.000 (0.1 mm/min ~15000.0mm/min)
Feed/rev.	0.001mm/r	F1~F500000 (0.001 mm/r ~500.000mm/r)
Thread lead F command	0.001mm	F1~F500.000 (0.001 mm ~500.000mm)
Thread lead E command	0.00001mm	E1~E9999999 (0.0001 mm ~99.99999mm)

2.4.5 Automatic Acceleration/Deceleration

A time constant automatically produces acceleration or deceleration to avoid the mechanical system trembles, during starting or stopping. So, the acceleration/deceleration is not considered when a programming is performed.

A pointed corner can not be machined due to automatic acceleration/deceleration. A dwell command (G04) should be inserted between two blocks if you machine a pointed corner.



If a dwell command is inserted, the actual tool path is consistent with programmed path. The faster the feedrate is and the longer the acceleration/deceleration time is, the more the corner error is.

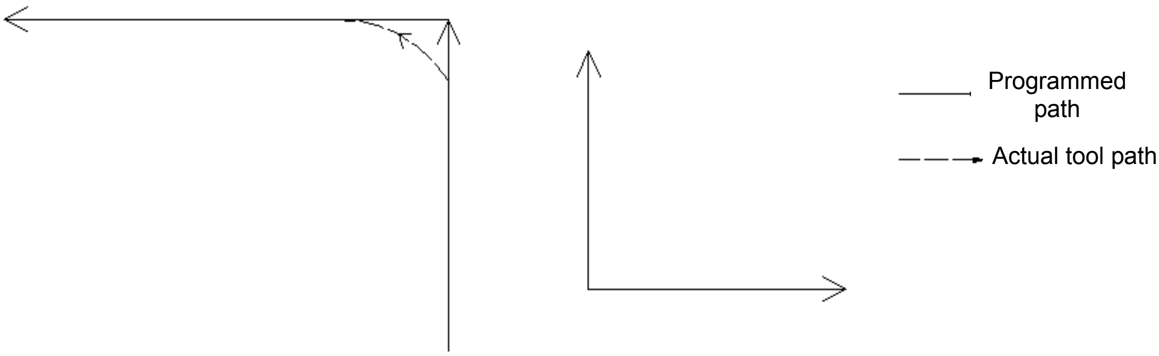
Note 1: NC processes between blocks, as shown below:

New block \ Previous block	Positioning	Cutting feed	Unmovable
Positioning	X	X	X
Cutting feed	X	O	X
Unmovable	X	X	X

X: Command speed is decelerated to zero; the next block is then performed again.

O: The following block is orderly performed; the feedrate is not much changer.

Note 2: Acceleration/deceleration is separately performed to each axis (X and Z axes), and the feedrate of each axis is changed between blocks, so, the actual tool path is inconsistent with programmed path. For example, a tool is only decelerated to feedrate along X axis in a block, here, the Z axis direction acceleration produces, the actual tool path as shown below:



The actual arc radius is less than the programmed arc radius in arc interpolation (see the D4.3 in appendix four). In the occasion of machine tool allowance, and In order to reduce this error, the small acceleration/deceleration time constant of cutting feed is employed.

2.5 Preparatory Function (G Function)

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

Type	Meaning
One-shot G code	The G code is effective only in the block in which it is specified.
Modal G code	The G code is effective after specifying once until another G code of the same group is replaced.

For example; G01 and G00 are modal G codes in group 01.

G01 X ________;

Z _______; G01 is effective in this range.

X_______ ;

G00 Z _______;

There are 3 series G codes. One is standard G code; the other is special G code B/C. Two selections of the codes are set by parameter #7.5 (GSP). This manual uses standard G code. Special

G code with same function and standard G code are indicated in the following table.

G90 and G91 are different. G90 indicates absolute command (The used X and Z are same with standard X and Z in G90 mode), and G91 indicates incremental command (The used X and Z are same with standard U and W in G91 mode) in special G codes. But, when the command is input in MDI mode, in G90/G91 mode, the address X and Z are indicated as absolute command, U and W are indicated as incremental command.

Special G codes are employed, the address U and W are same with standard time, and they are indicated as incremental move distance also, even if the G90 address is applied.

Standard G code	Special G code B	Special G code C	Group	Function
☆G00	☆G00	☆G00	01	Positioning (feedrate rapidly)
☆G01	☆G01	☆G01		Linear interpolation (cutting feed)
G02	G02	G02		Arc interpolation CW
G03	G03	G03		Arc interpolation CCW
G04	G04	G04	00	Dwell
G07	G07	G07		SIN curve control for feedrate
G10	G10	G10		Offset setting
G20	G20	G20	06	Input in inch
G21	G21	G21		Input in metric
G22	G22	G22	04	Memory stroke limit ON
G23	G23	G23		Memory stroke limit OFF
G27	G27	G27	00	Reference point return check
G28	G28	G28		Reference point return
G29	G29	G29		Returning from reference point
G30	G30	G30		Return to the 2nd reference point
G31	G31	G31		SKIP function
G32	G33	G33	01	Thread cutting in linear and taper
G34	G34	G34		Leading thread cutting changeably
G36	G36	G36	00	Automatic tool X compensation
G37	G37	G37		Automatic tool Z compensation
G40	G40	G40	07	Tool nose radius compensation cancel
G41	G41	G41		Tool nose radius compensation -Left
G42	G42	G42		Tool nose radius compensation -Right
G50	G92	G92	00	Coordinate system setting Spindle max. speed setting
G65	G65	G65		User macro program non-modal calling
G66	G66	G66	12	User macro program modal calling
G67	G67	G67		User macro program modal cancel
G68	G68	G68	13	Double tool-post image ON
G69	G69	G69		Double tool-post image OFF
G70	G70	G70	00	Finishing cycle
G71	G71	G71		Outer diameter roughing cycle

G72	G72	G74		Flat end roughing cycle
G73	G73	G75		Closed-loop cutting cycle
G74	G74	G76		Flat end groove cycle (Z axis)
G75	G75	G77		Outer diameter groove cycle (X axis)
G76	G76	G78		Thread cutting cycle
G90	G77	G20	01	Single canned cutting cycle A
G92	G78	G21		Thread cycle
G94	G79	G24		Single canned cutting cycle B
G96	G96	☆G96	02	Constant surface speed control
☆G97	☆G97	☆G97		Constant surface speed control cancel
☆G98	☆G94	☆G94	05	Feed/min.
☆G99	☆G95	☆G95		Feed/rev.
——	☆G90	☆G90	03	Absolute value command
——	☆G91	☆G91		Incremental value command

Note 1: A G codes with the sign of ☆ is the start G code of each group. Namely, A G code can be established after the resetting key is controlled, when the power is turned on or the parameter for described start G code is enabled. As for G22 and G23, G22 is selected when the power is turned on. G22 or G23 is a mode before resetting (one of them is effective before resetting).

As for the G00 and G01, G98 and G99 or G90 and G91; the G code in the initial state can be selected by setting parameter ＃7.

G20 or G21 is the state of before the power is turned off or the resetting key is controlled.

Note 2: G code in 00 groups is not modal. They are effective only in the programmed block.

Note 3: When a code does not display in the above table in block or a G code without a definition and a selection is specified, the alarm No. 010 occurs. But, G60 and G61 may not alarm.

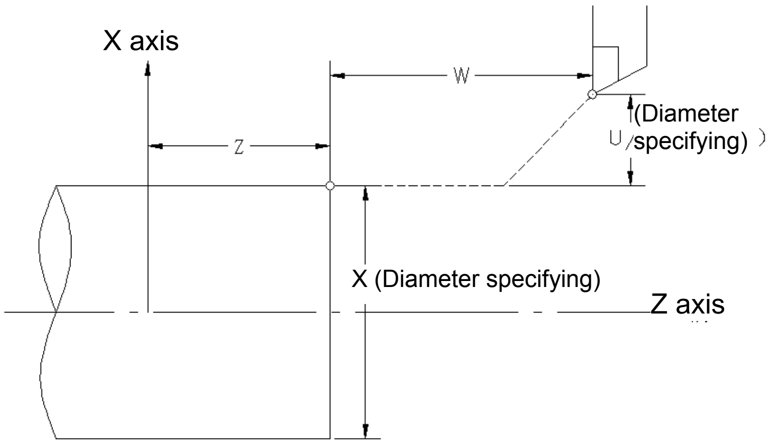
Note 4: Provide that some G codes are not share the same group; they can be described in a same block. When some G codes in the same group are described, the last one is effective.

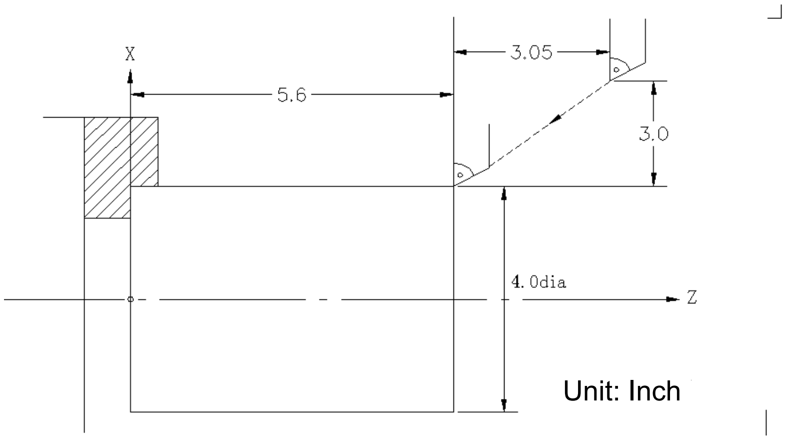
Note 5: G codes are shown one of them in terms of group.

2.5.1 Positioning（G00）

Function: Tool in the position of coordinate position (X, Z) or from the current position is rapidly traversed in terms of the specified distance along each axis separately.

Format: G00 X（U）—Z（W）—;





(Diameter programming) G00 X4.0 Z5.6; or G00 U-6.0 W-3.05;

Note 1: G00 rapid traverse rate of X, Z axis can be separately set by modifying parameter ＃92 and ＃93 by the machine tool builder . Rapid traverse rate can not specified by address F.

Tool is accelerated in the start of block till to the predictive speed in the mode of G00 positioning. And it is decelerated to 0 till to the end of block. In-position check can be set according to the parameter #6.5 (CINP), ensure that the next block executes after it is in the mode of in-position.

Note 2: "In-position" means that the feed axis motor reaches the range for specifying end position (this range is determined by setting parameter #70 and #71 according to the actual need by machine tool builder).

2.5.2 Linear Interpolation（**G01**）

Function: G01 Linear interpolation moves the tool to the position of X, Z at the speed specified by F in workpiece system, or, the current position moves to the position specified with U, W values.

Format: G01 X（U）—— Z（W）—— F——;

（Diameter programming）　G01 X4.0 Z2.01 F2.0 or G01 U2.0 W-2.59 F2.0;

(See the figure)

Feedrate specified by address F is the speed which tool moves along linear.

If feederate override is 0, the feedrate is 0, and the diagnosis parameter #700.4 is 1.

Note 1: feedrate of each axis, see the following:

Feedrate along X axis: $Fx = \frac{Lx}{L} F$

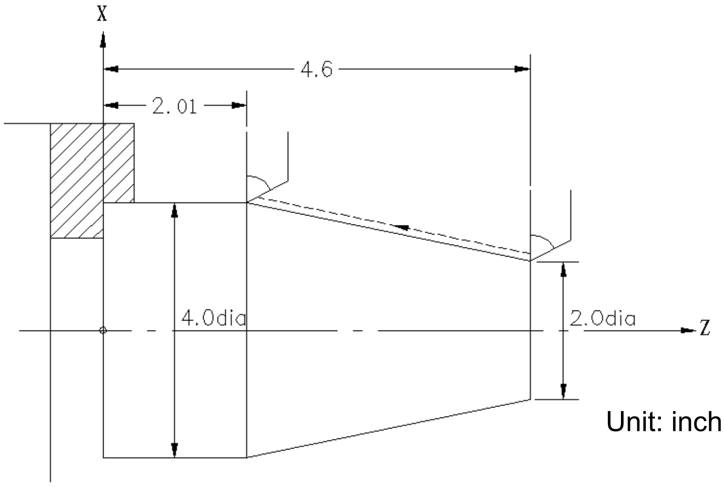
Feedrate along Z axis: $Fz = \frac{Lz}{L} F$

Thereinto: F: Feedrate

L: Movement distance

Lx: Movement distance along X axis

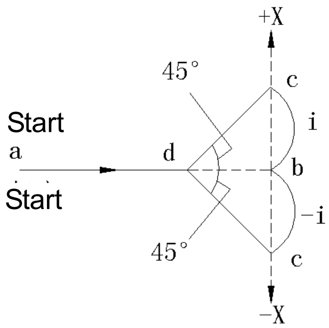
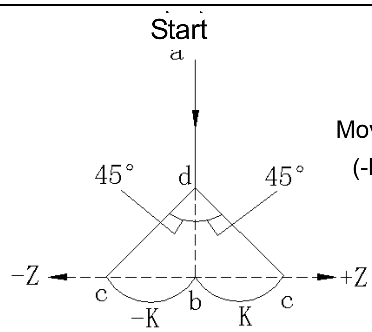
Lz: Movement distance along Z axis

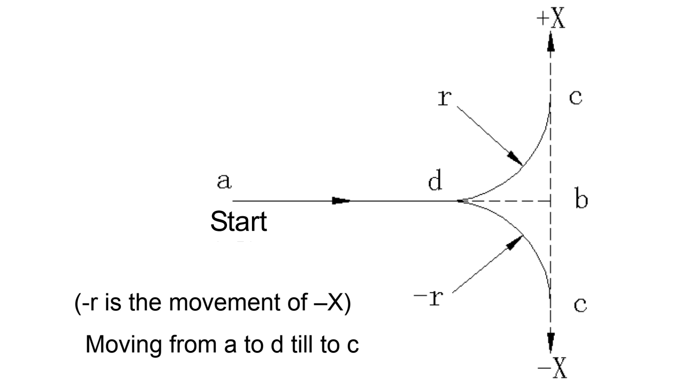
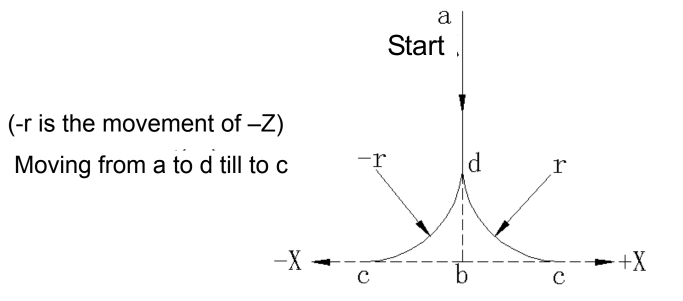


Unit: inch

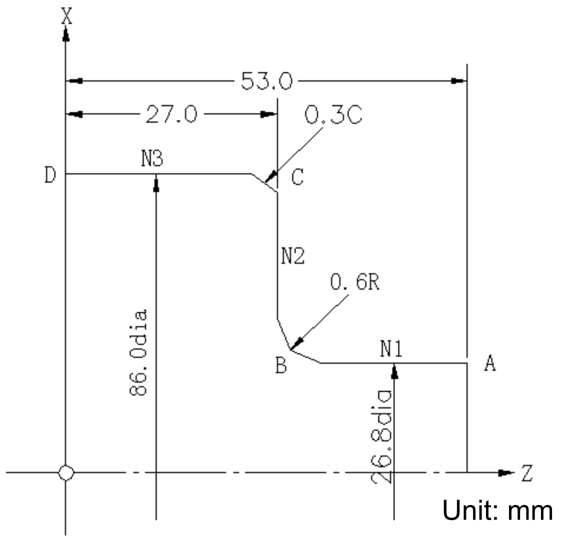
2.5.3 Chamfering and Corner R

The following chamfering and corner can be simply inserted between two blocks.

Item	Command	Tool movement
Chamfering Z→X	G01 Z（W）b I±i; ↑ In the right figure, tool moves to point b is specified with incremental or absolute command	 Moving from a to d till to c (-i is the movement of –X) Moving from a to d till c (-i is the movement of –X)
Chamfering X→Z	G01 X（U）b K±k; ↑ In the right figure, tool moves to point b is specified with incremental or absolute command	 Moving from a to d till to c (-k is the movement of –Z)

Corner R Z→X	G01 Z（W）<u>b</u>　R±r； ↑ In the right figure, tool moves to point b is specified with incremental or absolute command	
Corner R X→Z	G01 X（U）<u>b</u>　R±r； ↑ In the right figure, tool moves to point b is specified with incremental or absolute command	

I and K are always specified with the radius



（Diameter programming）

N0　G50 X26.8 Z53.0；　（point A）

N1　G1 Z27.0 R0.6 F100；（point B）

N2　X86.0 K-0.3；　　　（point C）

N3　Z0；　　　　　　　（point D）

Note 1: As for chamfering or corner R should be specified with G01 move along X or Z axis. Next block should vertical to the single command of X or Z axis.

Note 2: Alarm may generate in the following conditions.

1) When I or K or R is employed, X and Z axes are simultaneously specified with G01 mode. (Alarm No.054)

2) Two of I, K and R in same block are specified with G01 mode. (Alarm No. 053)

3) X and I or Z and K are simultaneously specified with G01 mode. (Alarm No.056)

4) In the block, the chamfering or corner R is specified, the movement distance along X and Z are less than the chamfering or corner R. (Alarm no.055)

5) The block which after a block is specified with chamfering or corner R, its movement without specifying is vertical to G01 of the previous block. (Alarm no.G51)

6) In the block, the chamfering or corner R is specified, its movement direction is incorrect (Alarm No.052).

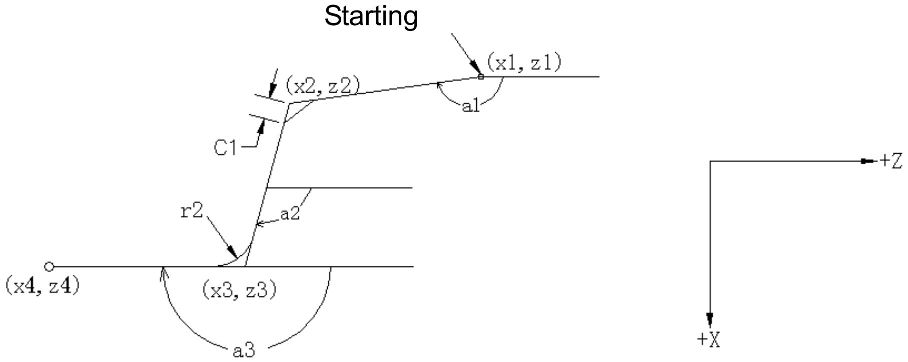
Note 3: In single block, the tool moves to the chamfering and corner R position which have finish executing the command in the block, and stops.

Note 4: Chamfering and corner R can not be used for the block of thread cutting.

2.5.4 Figure Dimension Input

The linear angle such as chamfering, corner R and other dimension value on the machine figure can be directly entered when programming. Furthermore, chamfering and corner can be inserted between the linear of random angle.

For example:



X（x2）Z（z2），C（c1）；

X（x3）Z（z3），R（r2）；

X（x4）Z（z4）；

or

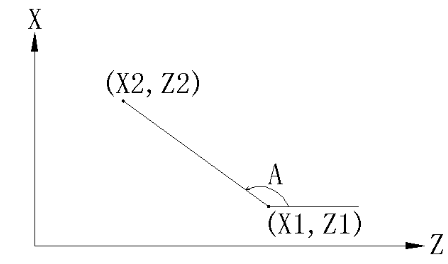
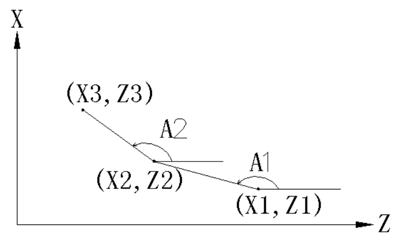
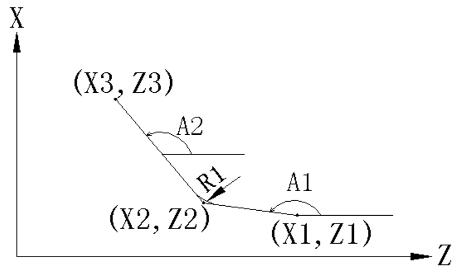
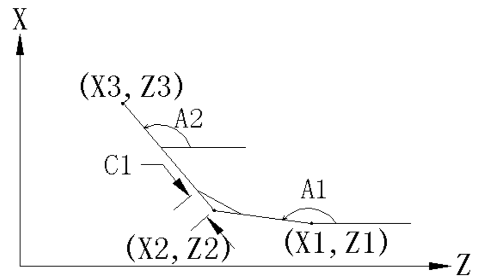
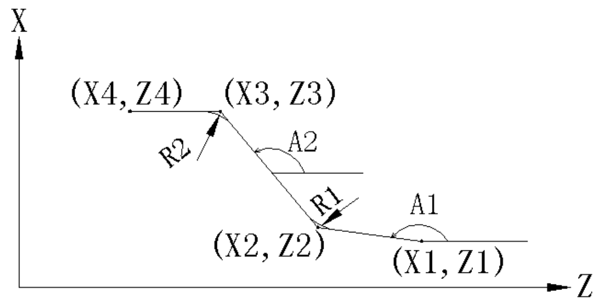
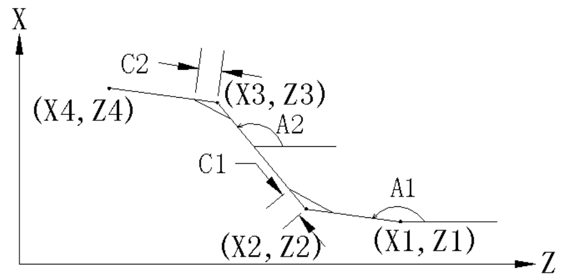
A（a1），C（c1）；

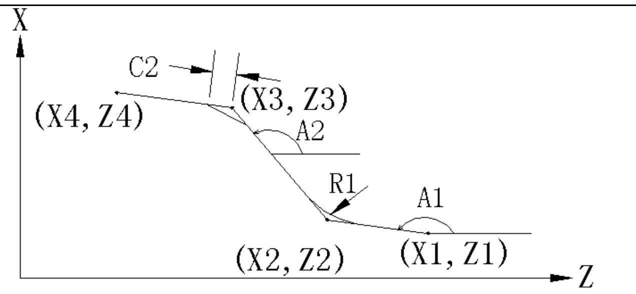
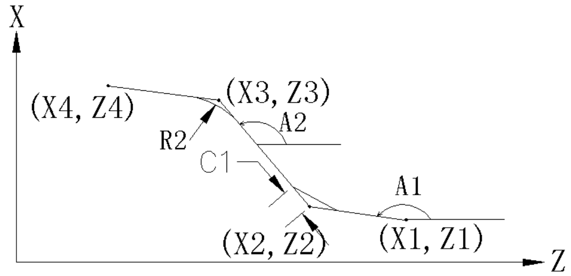
X（x3）Z（z3）A（a2），R（r2）；

X（x4）Z（z4）；

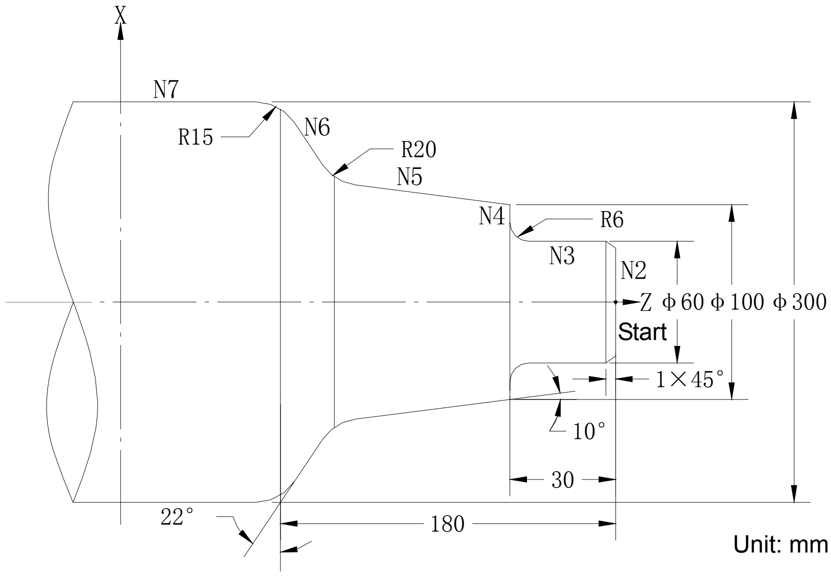
Note 1: A straight line is specified, it is necessary to specify one or two of X (U), Z (W) and A. A straight line should be specified at the beginning of next block if only one is specified. When chamfering and corner are specified, they should be separated by “,”, such as: C_, R_.

	Command	Tool movement

1	X_2____(Z_2____)A____；	
2	A_1____； X_3____Z_3____A_2____；	
3	X_2____Z_2____，R_1____； X_3____Z_3____； or A_1____，R_1____； X_3____Z_3____A_2____；	
4	X_2____Z_2____，C_1____； X_3____Z_3____； or A_1____，C_1____； X_3____Z_3____A_2____；	
5	X_2____Z_2____，R_1____； X_3____Z_3____，R_2____； X_4____Z_4____； or A_1____，R_1____； X_3____Z_3____A_2___，R_2____； X_4____Z_4____；	
6	X_2____Z_2____，C_1____； X_3____Z_3____，C_2____； X_4____Z_4____； or A_1____，C_1____； X_3____Z_3____A_2___，C_2____； X_4____Z_4____；	

7	X_2____Z_2____，R_1____； X_3____Z_3____，C_2____； X_4____Z_4____； or A_1____，R_1____； X_3____Z_3____A_2___，C_2____； X_4____Z_4____；	
8	X_2____Z_2____，C_1____； X_3____Z_3____，R_2____； X_4____Z_4____； or A_1____，C_1____； X_3____Z_3____A_2___，R_2____； X_4____Z_4____；	

(Program example) specifying diameter, input in metric



N001 G50 X0.0 Z0.0；

N002 G01 X60.0 A90.0，C1.0 F80；

N003 Z-30.0 A180.0，R6.0；

N004 X100.0 A90.0；

N005 A170.0，R20.0；

N006 X300.0 Z-180.0 A112.0，R15.0；

N007 Z-230.0 A180.0；

Note 2: The command uses the figure dimension input directly only when it runs in Auto mode.

Note 3: In the block, the figure dimension is directly entered the command, or between two blocks which serial figure dimension is entered directly, the following G codes can not be employed.

① **G codes of 00 group (Command other than G04)**

② **G02, G03, G90, G92 and G94**

Note 4: Corner R can not be inserted during thread cutting.

Note 5: Chamfering of 2.5.3, command of corner R, the chamfering of figure dimension directly input and corner R can not be employed simultaneously.

Note 6: The edge command of figure dimension enters directly, if the end point of previous block is determined by figure dimension of next block, single block can not be executed, however, the feed hold can be ran during the previous block execution.

Note 7: In the following block, the delimitation angle of intersection calculation is performed to ±1°. (The movement value gained by using intersection point calculation is over.)

X（u）——A——; (If an angle A is specified within 0±1°. Alarm may be generated within 180°±1°)

Z（w）——A——; (If an angle A is specified within 90±1°. Alarm may be generated within 270°±1°).

Note 8: The angle difference between two straight lines is less than ±1°, the alarm occurs when intersection point calculation is performed.

Note 9: Chamfering or corner R is effective when the angle between two straight lines is less than ±1°.

Note 10: A next block of that the block is only specified by the angle, the seat command (absolute value) and angle command should be specified simultaneously.

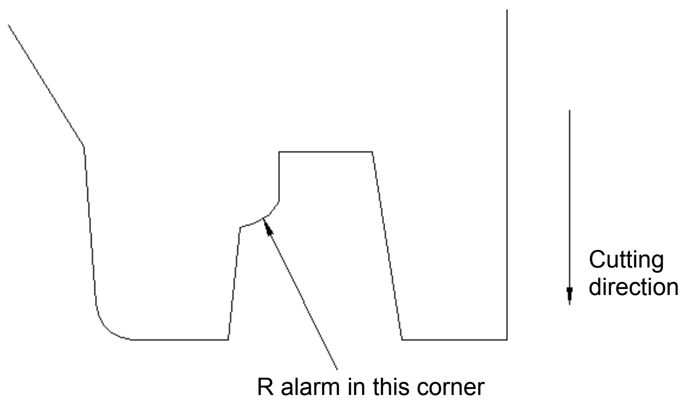
For example:

N1 X——A——, R——; As for N3 block, the coordinate and angle commands are simultaneously specified.

N2 A——;

N3 X——Z——A——;

Note 11: As for the II type of complex canned cycle (G71, G72), in the direction of cutting, chamfering or corner of intersection point between blocks can not be performed. If G70 is used to perform the block with figure dimension input directly command, No.066 alarm occurs.



2.5.5 Arc Interpolation（G02，G03）

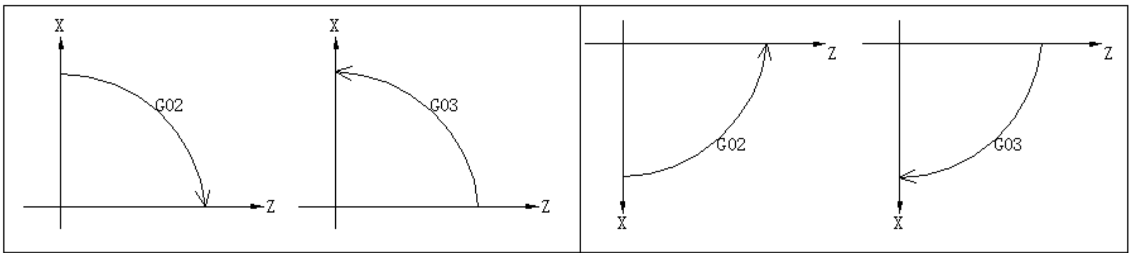
Function: The following command indicate the tool movement along the arc. The commands are shown below:

Format：G02 / G03 X(U)___ Z(W)___ { I____ K____ / R ____ } F____

Content		Command	Meaning
1	Revolution direction	G02	CW
		G03	CCW
2	End point position	X，Z	End position in workpiece coordinate system
	Distance to end point	U，W	Distance from start to end point
3	Distance from start to center	I，K	Distance from start to center (Radius specifying)
	Arc radius	R	Arc radius, arc within 180° (Radius specifying)

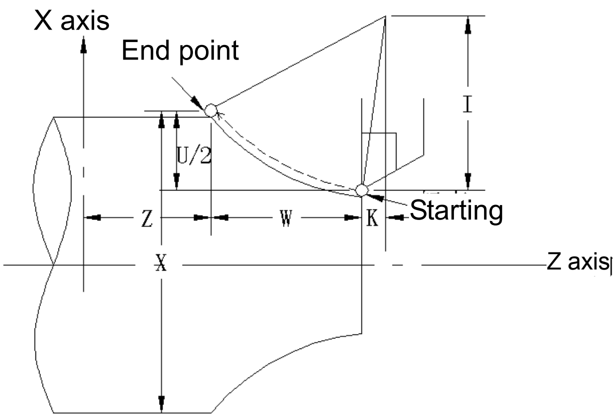
4	Feedrate	F	Feedrate along arc

The direction of CW or CCW applies when the coordinate system is in right or left hand.

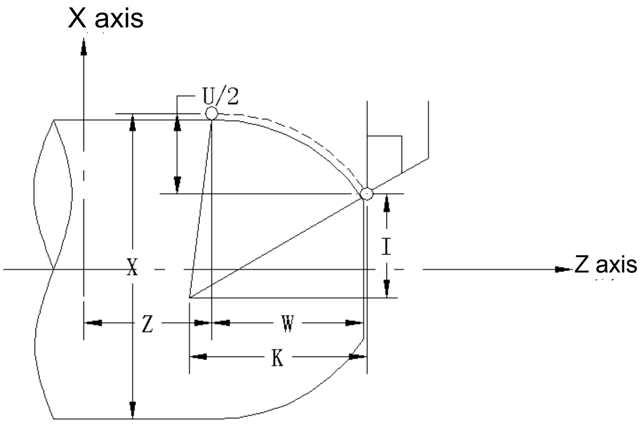


Right coordinate system	Left coordinate system

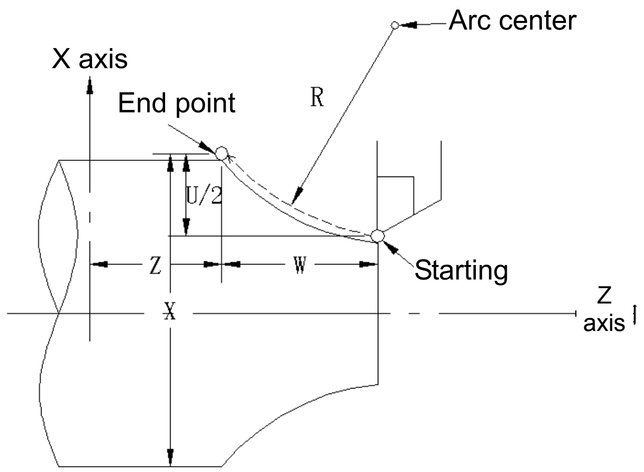
G02 X（U）——Z（W）——I——K——F——; (Diameter programming)



G03 X（U）——Z（W）——I——K——F——; (Diameter specifying)



G02 X（U）——Z（W）——R——F——; (Diameter specifying)



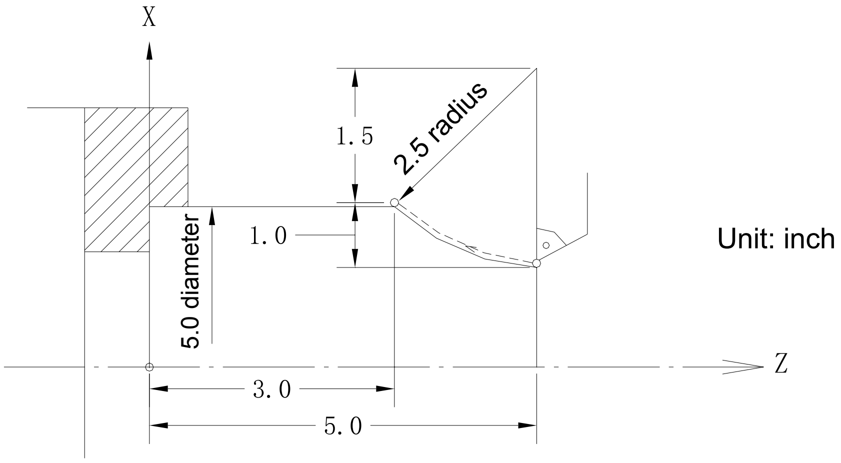
(Diameter programming)

G02 X5.0 Z3.0 I2.5 F0.03;

Or G02 U2.0 W-2.0 I2.5 F0.03;

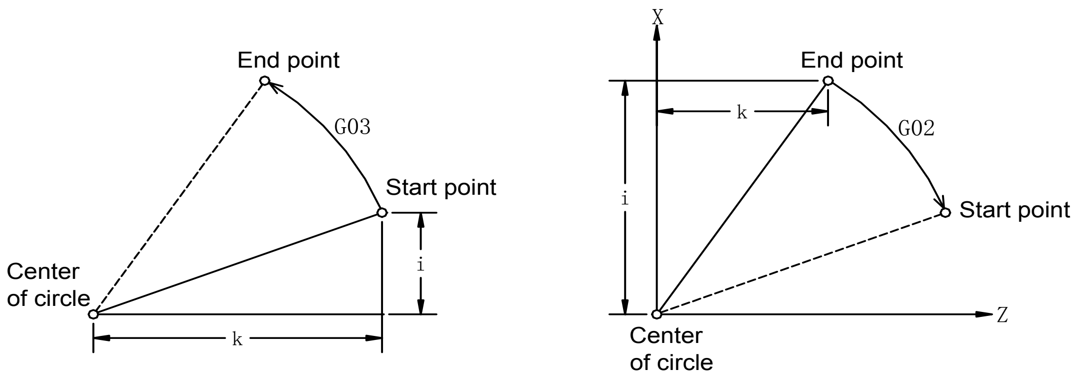
or G02 X5.0 Z3.0 R2.5 F0.03;

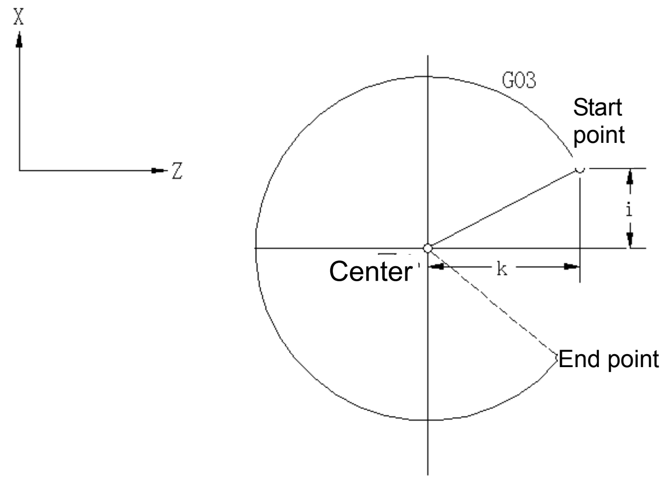
or G02 U2.0 W-2.0 R2.5 F0.03;



Note 1: This word can be omitted, when I or K is zero.

Note 2: I and K separately specified the distance from the start to center along X and Z axes. The sign should be considered.





Note 3: In programming, when the X, Z, U and W are all omitted, or U and W are zero or the position specified by X and Z are same with the starting point, the center is indicated by address I and (or) K, that is a round of 360° is specified.

For example: G02 I——;

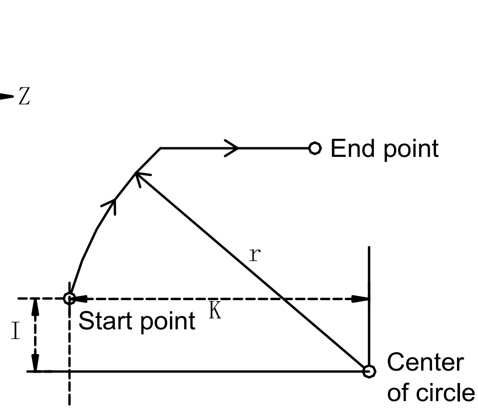
Note 4: An arc within 180° can be specified, when address R is employed in programming, and it is can not be specified when the arc is more than 180°.

So: G02 R——; and G03 R——; are not specify the whole round, but arc of 0° does. So, the tool not moves.

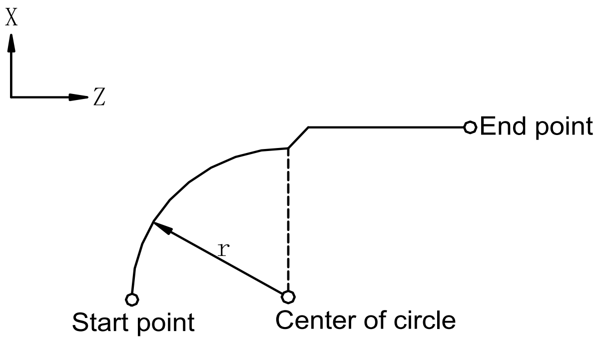
Note 5: When address R value specified is less than the half distance from start to end point（including R=0）, R is omitted, and a half round is generated. If R is negative, No.023 alarm occurs.

Note 6: When programming with I and K, and the end point is not on the arc, the tool moves in terms of the following figure.

(I) In arc command, end point of two axes is coordinate, one of them is on the arc, after tool reaches the coordinate value of the axis end, move directly to another axis end coordinate values by linear mode.



(II) In arc command block, when end point of two axes are not on the arc, the arc path formed by corresponding I and K value is firstly performed, and then move to the end point position with nonlinear movement, the results are shown below:



Note 7: Feedrate can be specified using address F in arc interpolation, see 2.4.2, the error of corresponding actual feedrate is within ±2%. The feedrate performs along the arc after tool nose compensation is executed.

Note 8: When I, K and R are specified simultaneously, R is enabled, but I and K are omitted.

Note 9: I and K are specified zero arc, tool straightly moves from start to end using G01 mode.

2.5.6　Feedrate for SIN Curve Control（G07）

Function: An axis in arc plane is not moved during arc cutting command, the interpolation performs only (it is regarded as imaginary axis) along the feedrate of axis, another feedrate of axis SIN curve change is formed.

Format：G07α0; (α is regarded as imaginary axis)

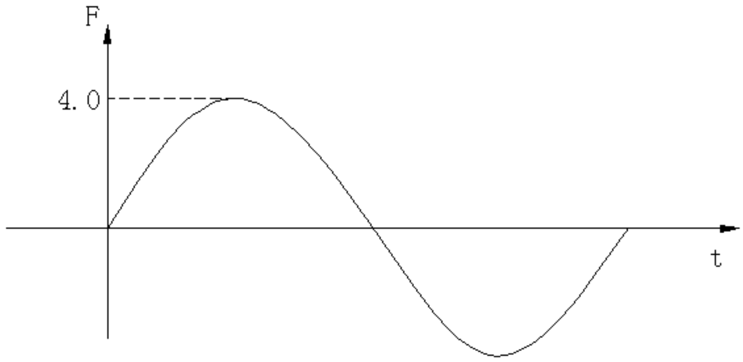
G07α1; (α is regarded as movement axis)

Specifying as follows:

G07 Z0; ……Z axis is set to an imaginary axis.

G02 X0 Z0 I10.0 F4; ……Feedrate of X axis changes with SIN curve.

G07 Z1; …… Cancel imaginary Z axis.



Note 1: The imaginary axis is only enabled for automatic operation. It is disabled for manual. That is movement.

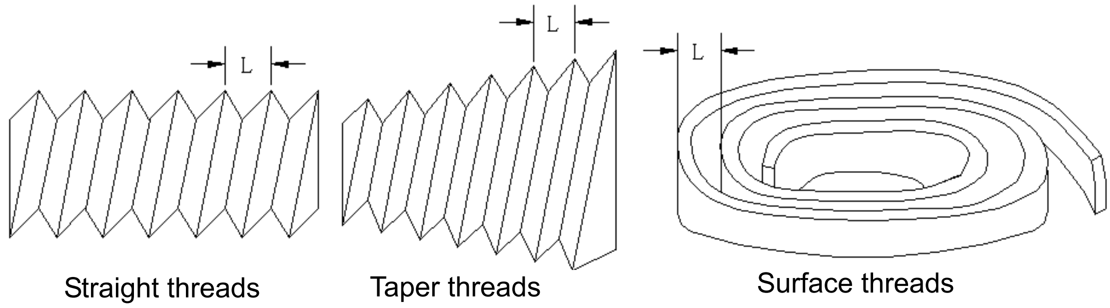
Note 2: Inter-lock, stroke limit and outer deceleration are also effective to imaginary axis.

Note 3: The MPG interruption insert is also effective for imaginary axis, this axis is executed as MPG interruption insert part. Diagnosis ＃805 and ＃806 are separately the insert amount of X and Z axis.

2.5.7 Thread Cutting（G32，G34）

2.5.7.1 Linear, Taper Thread Cutting（G32）

Function: The straight, taper and surface threads cutting can be performed by the command G32.



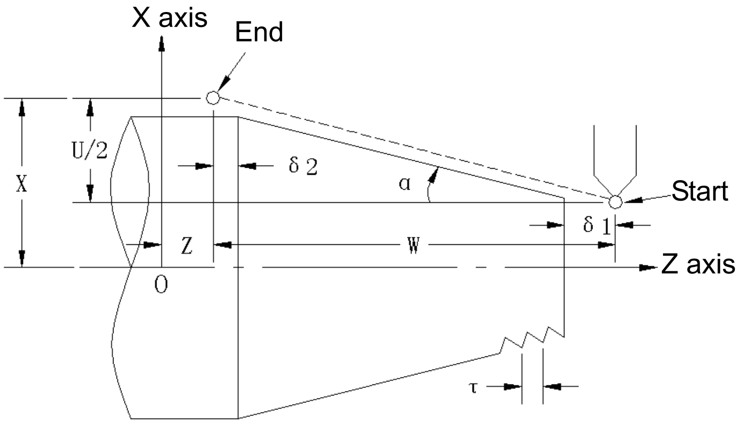
Values followed with F and E are directly led to perform the thread cutting. Thread cutting command as follows:

The coordinate position (X, Z) in the workpiece coordinate system is specified the end point or it is specified the distance from start to end points (U, W).

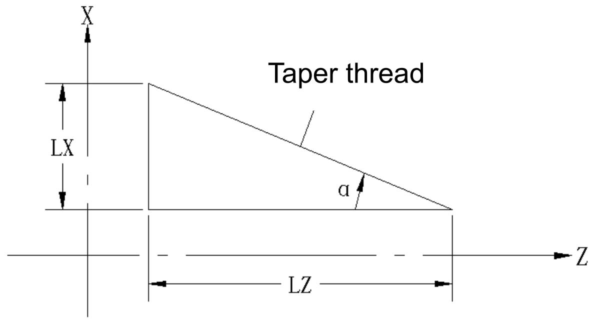
Format: G32 X（u）——Z（w）——F——；

G32 X（u）——Z（w）——E

Lead length



In general, thread cutting is repeated along the same tool path (including from rough cutting to finish cutting). Since thread cutting starts when the position coder mounted on the spindle outputs a 1-turn signal, so, thread cutting is started at a fixed point. 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.



The lead is LZ if α≤45°
The lead is LX if α> 45°

Thread lead is generally specified with a radius.

For example: Thread lead and command value…………command for F code.

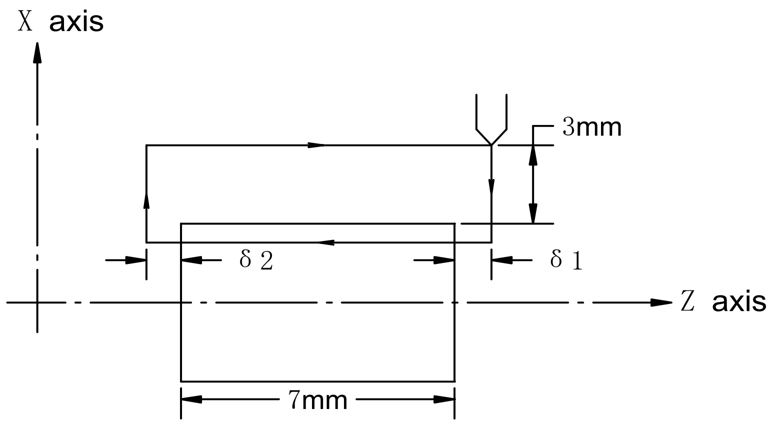
Input unit	F14	F5265
inch	0.0014inch	0.5265inch
mm	0.14mm	52.65mm

For example: Command value and thread lead…………E code

Input unit	E2346	E176534
inch	0.002346inch	0.176534inch
mm	0.2346mm	17.6534mm

Generally speaking, because of servo system delay, start and end points in thread cutting can not have correct value. In order to compensate this value, the specified thread length should be longer than the need one. See the Section 2.4.3.

For example: 5.5.1.1 Straight thread cutting



The following values are employed during programming:

Thread lead: 0.4 mm

δ_1 : 0.3 mm

δ_2 : 0.15 mm

Cutting depth: 0.1 mm (cutting twice)

(Input in inch, radius programming)

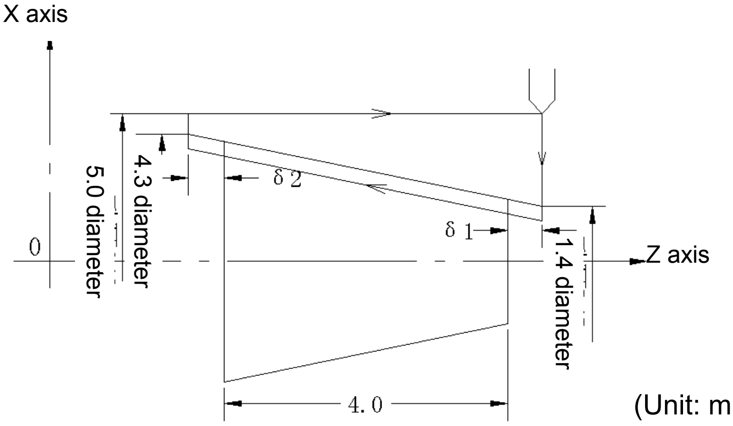
G00 U-3.1；
G32 W-11.5，F0.4；

G00 U 3.1；

W 11.5；
U-3.3; (Cut more 0.1 mm for the 2nd cutting)

G32 W-11.5；
G00 U-3.2；
W-11.5；

For example: 5.5.1.2 Taper thread cutting



(Unit: mm)

The following values are used for programming (Metric input, diameter programming)

Thread lead: it is 0.35 mm on Z axis direction

δ_1 ; 1.2 mm

δ_2 : 0.1 mm

The cutting depth along X axis direction is 0.1 inch (cutting twice)

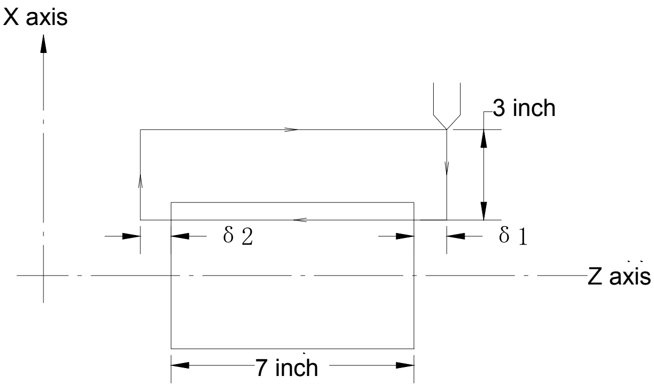
G00 X1.2；
G32 X4.1 W-5.3 F0.35；

G00 X5.0；

W5.3；
X1.0；
G32 X3.9 W-5.3；
G00 X5.0；
W5.3；

For example: 5.5.1.3 accurate thread cutting

Specify the thread lead can be accurate thread cut with E code (inch input, diameter programming)



Thread lead: 0.846667 inch

δ_1: 0.3 inch

δ_2 : 0.15 inch

Cutting depth: 0.1 inch (cutting twice)

G00 U-2.9；
G32 W-7.45 E0.846667 inch;

U2.9；
W7.45；
U-3.1; (Cut to 0.1 inch for 2nd cutting);

G32 W-7.45；
G00 U3.1；
W7.45；

Note 1: E code is only enabled when thread cutting.
Note 2: If E and F share the same block, the latter one is valid.
Note 3: Feedrate modification is disabled during thread cutting (it is fixed on 100%)
Note 4: whether the thread cutting is enabled or not in Dry run is set by parameter #8.0.

Note 5: It is very dangerous to stop thread cutting feed when the spindle not stops. So, the feedhold is disabled during thread cutting, if the feedhold is applied in cutting, the tool stops after the block of thread cutting is finish, it is the same as pressed the single block button. Therefore, when the FEED HOLD button on the machine operator panel is pressed, the feedhold indicator light is ON. And, when the tool finishes the current cutting and stops, The light is then OFF (in the sate of single block stop) .

Note 6: If the Feedhold key is pressed all the time in thread cutting, or pressed in the first block without specifying the thread cutting follows the thread cutting block, the tool stops at the block without specifying thread cutting.

Note 7: When thread cutting is performed in single block, after the first block without any specifying it is performed, tool stops.

Note 8: When the automatic mode is changed into manual (during thread cutting). It is the same as Note 6 that the serial feedhold buttons, the block start of thread cutting is absent, it is feedhold stops. But, when manual is changed into automatic mode, after the block without thread cutting is executed, it is stopped, see Note 5.

Note 9: When the previous block is the block of thread cutting, even the current block is thread cutting block, in the occasion of the detection signal is not waited, the thread cutting of current block is performed immediately.

G32Z—F—；
Z—; (The detection of each revolution is not performed before this block)
G32; (The block is thread cutting block)
Z—F—; (Same, the detection of 1-turn is not performed)

Note 10: Incorrect 0.01 thread part may occur between two blocks when the block of thread cutting is continued, namely, the speed changes within 0.01s. If command transform does not finished in time, when short thread is serially cut, more incorrect part may occur.

Note 11: Constant surface speed control is enabled while surface thread or taper thread is cutting, since the spindle speed has changed, the incorrect thread may be occurred. Therefore, the constant surface speed control can not be employed during thread cutting.

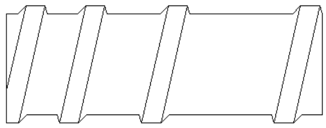
Note 12: Chamfering and corner R is not specified in the move block previous the thread cutting block.
Note 13: Chamfering or corner R can not be specified in a thread cutting block.
Note 14: Spindle speed modification is enabled in thread cutting mode, so, correct thread cutting can not be performed due to the servo system is delayed. The parameter #9.2 (SOVC) can be changed to make the spindle modification speed disabled in thread cutting.

Note 15: As for the G32, the thread cutting dwell is disabled.

When serial thread cutting selection is applied, it is possible that serial block cutting is performed and the correct thread part still retains during the block.

2.5.7.2　Variable Thread Lead Cutting (G34)

Function: Thread lead for each revolution is specified to added or reduced to perform the variable thread lead cutting.



Variable thread lead

K is specified the increase value or decrease value of each thread in the following commands.

Format: G34X(U)——Z(W)—— { F—— / E—— } K——;

An address other than K is the same as the straight/taper thread cutting in G32.

K values range can be employed:

Input in metric: ±0.0001 mm/r～±100.0000mm/r

Input in inch: ±0.000001 inch/r～±1.000000inch/r

E code can be employed only E6 digits selection is offered.

Note 1: As long as lead increase value or decrease value are over the command range, the alarm occurs (014)

Note 2: As for the G34, “Thread cutting dwell” is disabled.

2.5.7.3　Consecutive Thread Cutting

It is possible to specify the consecutive thread cutting block (G32, G34). Since a few pulses output of intersection of block is coincident with that of the next move block, the pulse distribution interruption time between two thread cutting blocks is eliminated. Therefore, multiple thread blocks can be specified continuously, and the helical may not be damaged even if changing the cutting depth in cycle thread cutting in a same position.

2.5.8　Auto Reference Position Return（**G27～G30**）

2.5.8.1　G27 Reference Position Return Check

Function: A point fixed on the machine is called reference position, and the tool can be returned

to reference position by the "manual reference position return".

G27 command is employed to determine whether the tool has reached to reference position, this reference position is specified in programming, if the reference point specified is not coincident with that of machine, No.092 alarm occurs.

Format: G27X（u）— Z(w)—;

When the above commands are employed, the tool moves to the specified position at the rapid traverse rate. If tool reaches the reference position, the indicator of reference position is ON. If only one axis reaches the reference position, the indicator of this axis's reference position return is ON. If the specified axis not reached there, the alarm occurs (No. 092)

Note 1: If an offset value is specified, a specified position reaches by G27 command will move an offset value. If a tool offset is not reach the reference position, the corresponding light is OFF. Usually, the offset should be erased before G27 is specified.

Note 2: The inch mechanical system is entered in metric, even if the tool programming position is offset 1μ from reference position, and the indicator light is also ON. The min. input value is less than the command incremental of mechanical system.

Note 3: G27 function just for the 1st reference position of machine.

2.5.8.2 G28 Automatic Reference Position Return

Format: G28 X（u）— Z（w）—；

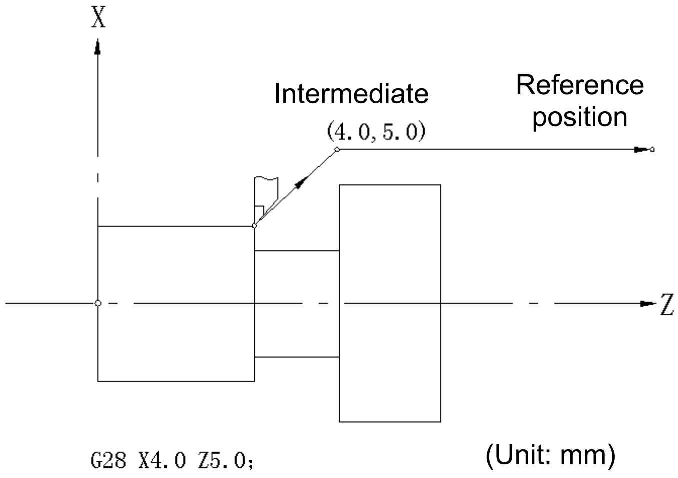
Function: Auto reference position return is specified by this command (for the specified axis). Positioning to the reference position is determined by the intermediate point specified by the X(u)_ Z(w).

Tool moves to intermediate point on all specified axes at rapid traverse rate after G28 block is employed, rapid traverse reference position return is then performed. Reference position return indicator light is ON, if it is not on the state of machine lock.

Positioning to the intermediate point or reference position, it is same effective to position with G00.

In general, this command is used for automatic tool change (ATC). Therefore, for safety, tool nose compensation and tool offset should be cancelled before executing this command.

Note 1: Before G28 command is specified, if a manual reference position return is not performed, after the power is turned on, the movement after the intermediate is same with manual reference position return. In this case, the direction from intermediate return selected by parameter #12 is equal to reference position return.



Note 2: The coordinate values of intermediate of G28 movement command are written down, in other word, there is no specification axis in G28 block, the current coordinate value of this axis is regarded as intermediate coordinate value of these axes.

For example:

N1 X1.0 Z2.0;
N2 G28 X4.0; intermediate point (4.0, Z2.0)

N3 X1.0 Z2.0
N3 G28 Z6.0; intermediate point (1.0, 6.0)

2.5.8.3 G29 Automatic Return from Reference Position

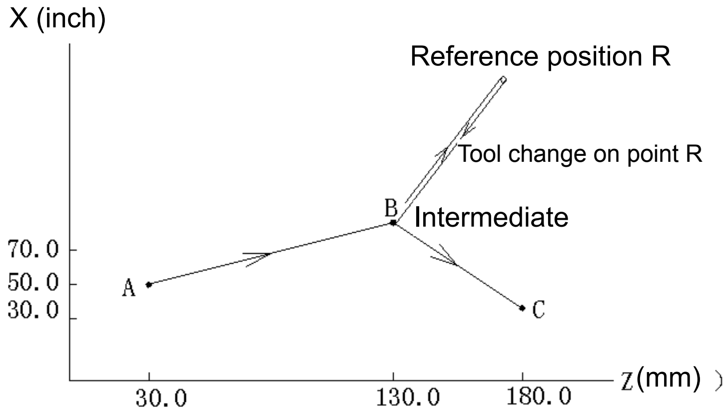
Format: G29 X（u）— Z（w）—;

This command is used for tool moves to specified position by intermediate. In general, this command is followed with G28 or G30 command.

In incremental command, the corresponding intermediate distance should be specified.

In G29 block, all specified axes can be moved to intermediate point specified by the previous G28 command at the rapid traverse rate, and then to the specified point.

Example for G28 and G29



（Diameter programming）

G28 U40.0 W100.0; (Program from A to B)

T0202;

G29 U-40.0 W50.0; (from the intermediate point specified by G28 to the program from B to C)

The above-mentioned indicates, programmer is not need compute the actual distance from intermediate to reference position.

2.5.8.4 G30 Returns the 2nd Reference Position

Function: The command is employed on specified axis to move the tool to the 2nd reference position.

Format: G30 X（u） Z（w）;

The 2nd reference position is set by the distance of 1st reference position with parameter.

This function is used for 1st reference position return but not for the 1st reference position return, and it is same as G28 returns to the reference position. A G29 command applied along with the G30 can make the tool position to the situation through the intermediate point set by G30, and its movement is the same as command G28 followed with G29.

Usually, G30 uses the different occasions for the tool position and reference position in the automatic tool change (ATC).

Note: Before G30 command, if the system is power on just now or the emergency stop is released or 4 alarm is cancelled just now, Manual reference point return or auto reference point return (G28) should be at least performed once .**

2.5.9 Dwell（G04）

Format: G04X (t); or G04U (t); or G04P (t);

Function: Any kinds of these commands can be used for dwell. Before starting the next block, wait for (t) ms after the previous is executed.

The max. command time is 99999.999s. The time error is within 16ms.

For example: Dwell 2.5 sec.

G04X2.5; or G04U2.5; or G04P2500;

Note 1: Address P can not be specified by the decimal point.
Note 2: The dwell starts in the following two cases.
Which one is enabled is determined by CINP.
1) After the previous block is turned to 0;
2) After the machine tool reaches command value (after positioning)
Note 3: When input in inch, if U or X is before G04, the dwell time is 10 times of standard format one.
Example: U1.0000 G04 (when input in inch, the dwell time is 10s)
Note 4: When G04 shares a same block with other movement commands, G04 is valid, and the others are not performed.
Example: G01 X10.0 G04 F100 (When input in metric, dwell is performed 10s, linear movement G01 is not performed)

2.5.10 Coordinate System Setting（G50）

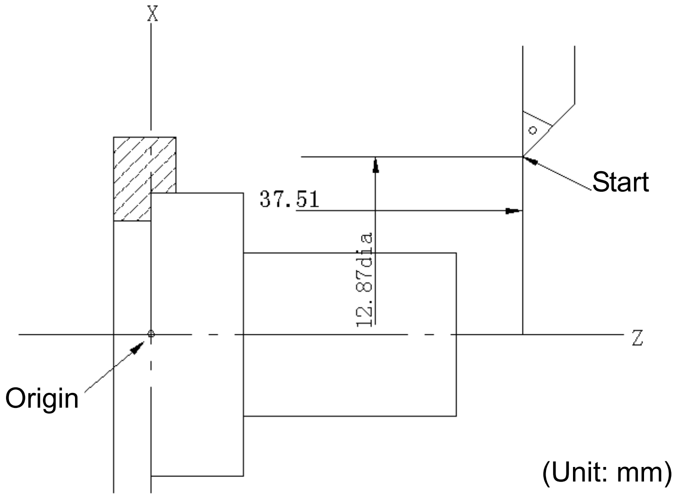
2.5.10.1 Command for Coordinate System Setting

The following command determines coordinate system:

Format: G50 X（u） Z（w）;

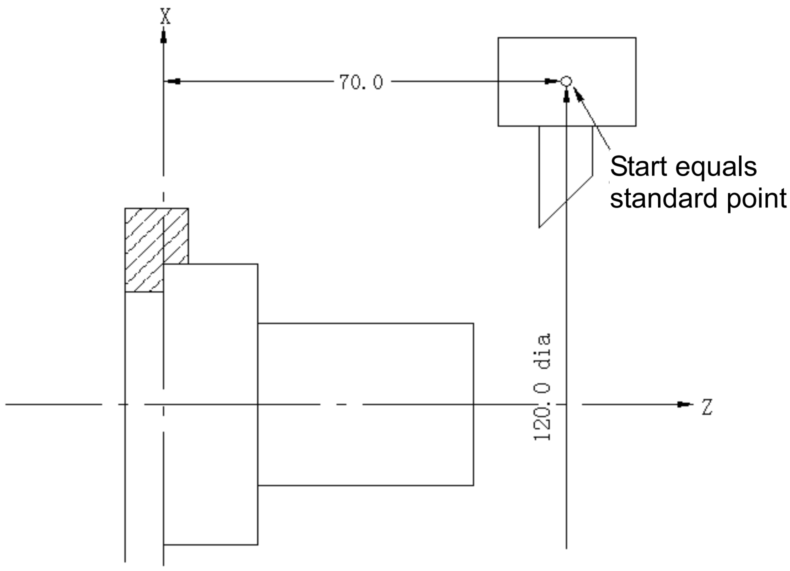
Function: The coordinate system of distance from current tool position to the origin of coordinate system is setup after the command is executed. This coordinate system is called workpiece coordinate system. Once it is setup, the following absolute command is regarded as the coordinate value within this coordinate system. The workpiece coordinate system is used for workpiece machining.

The X is diameter value when programming with diameter.



G50X12.87Z37.51; (Diameter programming)

In above program, G50 is used for ensuring the tool nose position is coincident with program starting point, and it must be defined at the beginning of block.



G50 X120.0 Z70.0; (Diameter programming)

As the above mentioned shown, some base point on tool post is consistent with the starting point of program, and coordinate system should be set at the beginning of program by G50.

In order to make the tool nose move along the programmed path, the position difference from tool nose to the base point must be compensated with tool offset.

Note 1: When a coordinate system is specified by G50 in offset mode, the specified position is tool coordinate value which is including the tool offset.

Note 2: Tool nose compensation specified by G50 is temporarily erased

Note 3: if G50 is followed by 01 group modal command, execute the G50 and change the current modal of 01 group to be command modal.

Example: G50G00X20.Z30, execute G50 coordinate setting and change the current modal of 01 group to be G00.

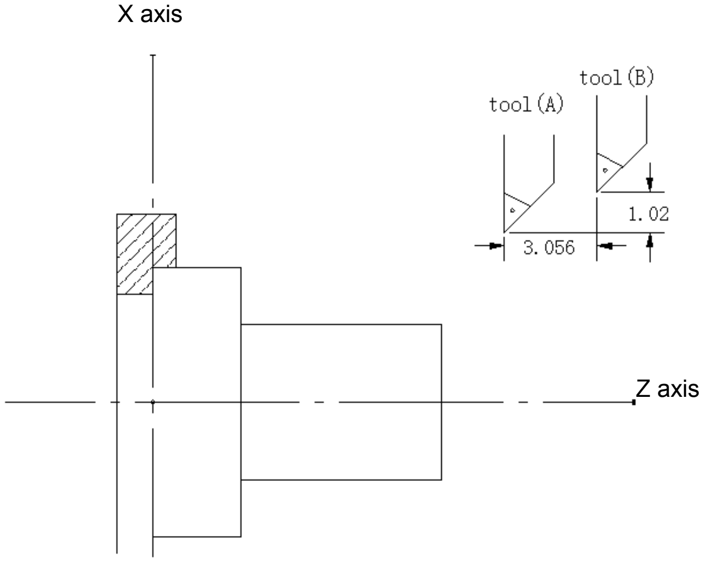
2.5.10.2 Coordinate System Movement

Workpiece coordinate system can be moved using the following commands.

G50 U（u）W（w）;

When the old coordinate value is X and Z axes, and the tool nose coordinate value is (X+U) and (Z+W) in new coordinate system.

When programming by diameter, the values of X and U are diameter.



The workpiece coordinate system with same zero points can be set by the following G50 command when the tool (B) is replaced of (A) in the above figure.

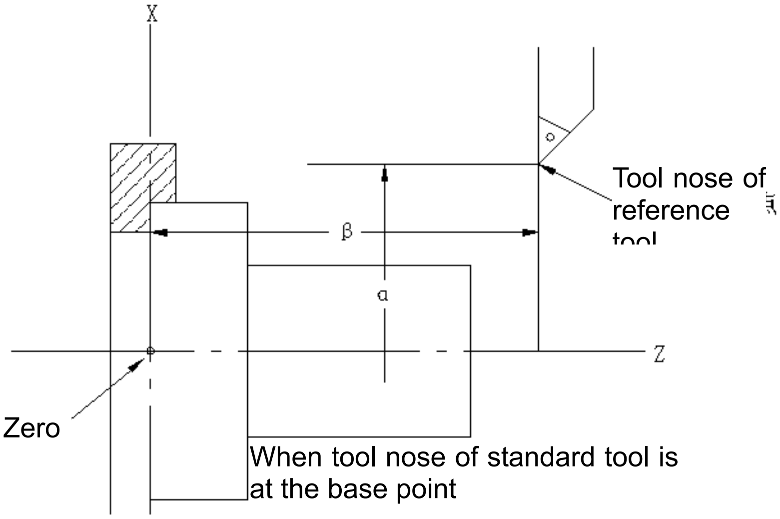
G50 U2.04 W3.056;（Diameter programming）

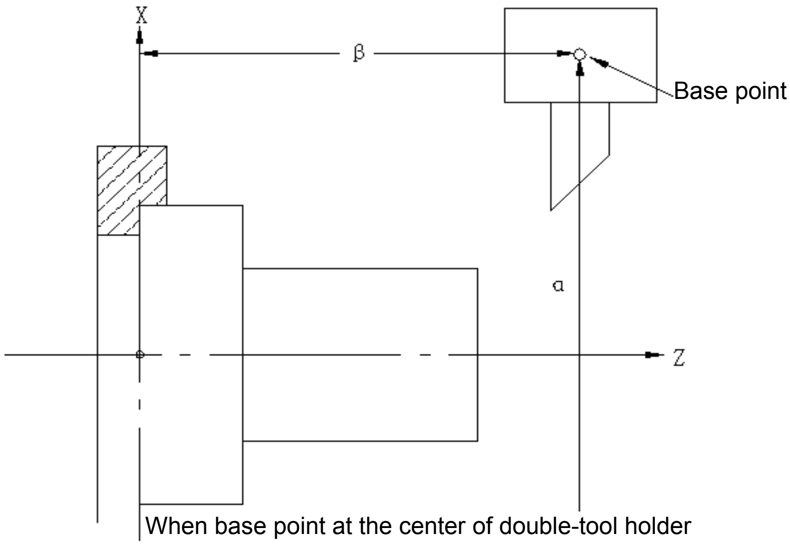
2.5.10.3 Automatic Coordinate System Setting

If the automatic coordinate system setting is valid with parameter (whether the automatic coordinate system setting is valid is set by parameters #309 APX and APZ). And the workpiece coordinate system is automatically set after manual reference point returns. In this case, coordinate values α and β should be set by No.375 and No.376 parameter beforehand.

If above parameters are set beforehand, when manual reference point return is completed, coordinate values α and β in the coordinate system of tool nose of reference tool or some base point of tool post is automatically set. It is the same as the tool is specified by G50 Xα Zβ at reference position.

G50 Xα Zβ;

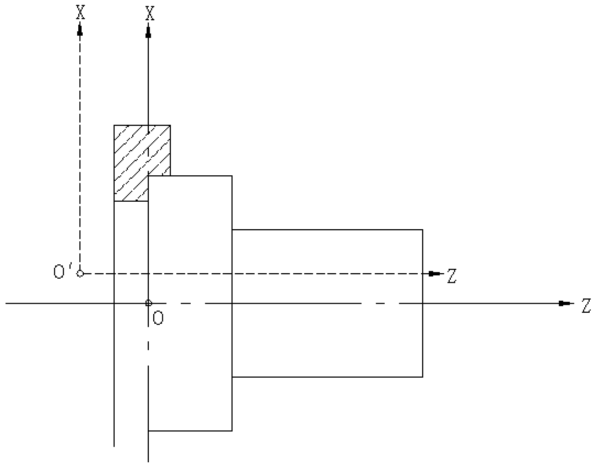




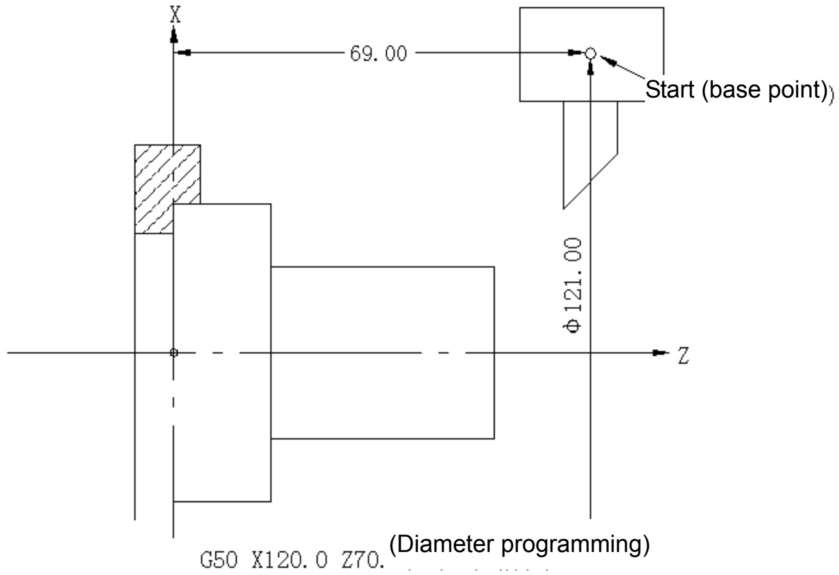
2.5.10.4 Workpiece Coordinate System Offset

If the workpiece coordinate system specified with G50 or coordinate system set by automatic coordinate system is different from the programming coordinate system, well then the coordinate system offset can be set beforehead to coincide with the programming coordinate system.

Offset values can be set in offset number 00 using offset memory. The setting mode is same as tool position offset value.



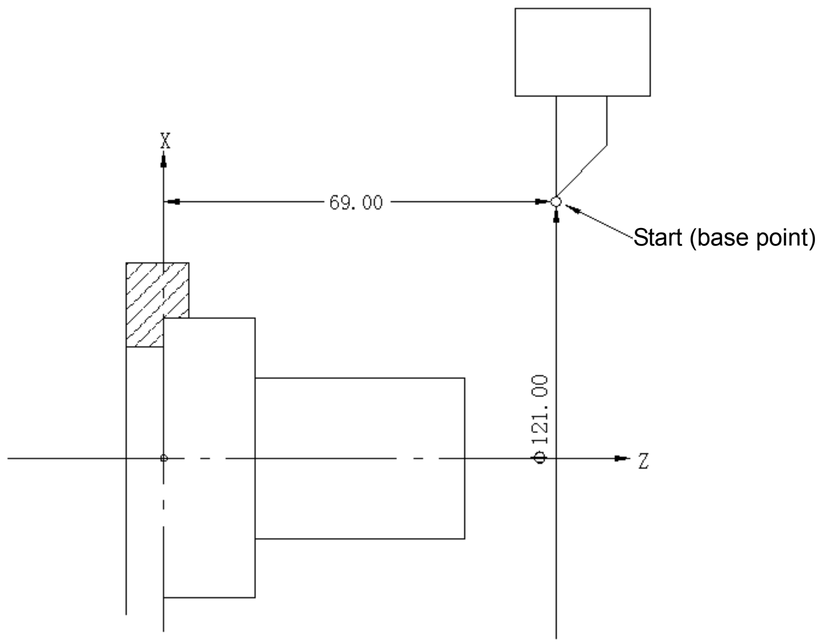
Movement value from O' to O is set by using no. OO offset memory



When base point position (diameter value) in X direction is 121mm, and is 69 mm in Z direction, G50 command is performed, see the following:

G50 X120.0 Z70.0；

Therefore, the actual position can be gained by moving the current workpiece -1.00mm along X axis direction, and 1.0mm along Z axis direction.



As the above figure shown, if the standard point is at the too nose, coordinate system is moved and set the needed coordinate system by using the same method. The move value along X axis is -1.0mm, and 1.0mm along Z axis. Then execute G50X120.0Z70.0 again, and the workpiece coordinate system will be set correctly.

When setting the movement value, the explanation is the most simple and convenient in section 5.10.5.

Note 1: Workpiece coordinate system is immediately moved after offset value is set.

Note 2: If coordinate system is set again with G50 after offset setting, movement value is omitted and to execute the workpiece set by G50.

(Example）: If the following command are specified, the coordinate value of current base point is X=100.0, Z=80.0 regardless of how the previous movement value is set.

G50 X100.0 Z80.0;

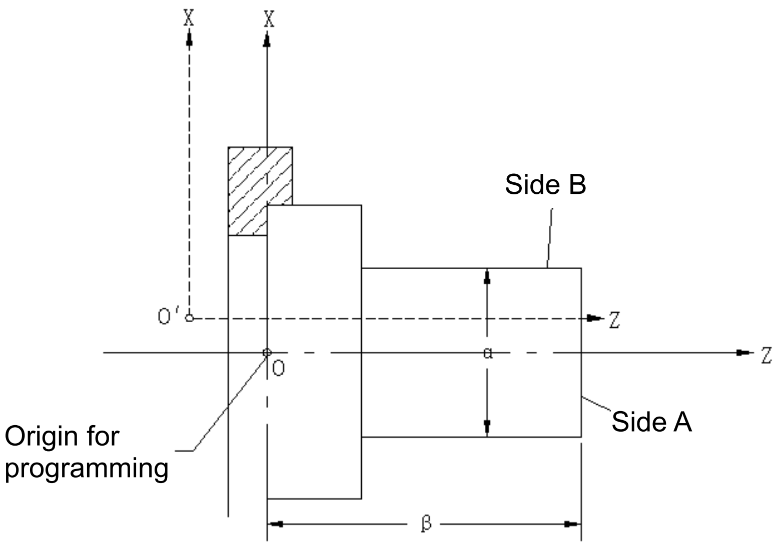
Note 3: After the offset is set, if manual reference point return or automatic coordinate system setting is performed, the offset works and the coordinate system is then immediately moved.

2.5.10.5 Workpiece Coordinate System Offset Value Input Directly

Workpiece coordinate system is not consistent with the coordinate system set by G50 or the automatic coordinate system, so, moving the coordinate system by inputting the offset value directly is performed when programming.

X Z—0 : The coordinate system is employed when programming.

X Z—0 ´ : Workpiece coordinate system for actual setting



(1) Manually cutting workpiece is performed along side A with reference tool.

(2) After INPUT button is pressed on machine operator panel, tool retracts and spindle rotation stops.

(3) In exterior page, input offset number N100, then input Z to store the measure distance β in above figure into offset memory.

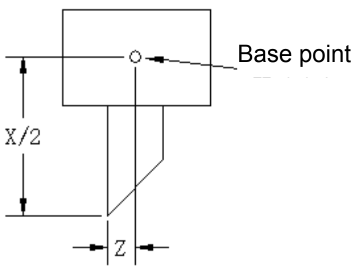
(4) Manually cut workpiece along side B.

(5) Tool retracts and spindle rotation stops after the position record button is pressed when the X axis does not move.

(6) In contour page, input offset number N100, then input X to store the measure distance **α** into offset memory.

From 0 to 0´, the movement value is automatically set into the offset memory of which the offset number is 00, and the workpiece is immediately moved.

If the offset value of reference tool is 0. When the reference tool nose is at the origin, the tool nose coordinate value in workpeice system (X=0.0, Z=0.0) is set.



However, the tool offset function is effective if tool offset value is set according to the above figure. As for the movement of workpiece coordinate system, the measure value input is directly employed. The base point workpiece coordinate systems of coordinate values X=0.0 and Z=0.0 (when the base point value is set at the origin) are established on the workpiece coordinate system.

Note: The measured value of X axis is employed with diameter value.

2.5.10.6 Double-tool Post Mirror Image (G68, G69)

G68 code is used to generate the mirror image of X axis.

G code	Meaning
G68	X mirror image ON
G69	Mirror image OFF

As for double tool post, the program is as follows:

G50U—— (Distance between double-tool post);

G68；

The following double-tool post machine is performed, and the program is invariable, it is same as the original tool post is machined the program.

2.5.11 Inch/ Metric Conversion（**G20，G21**）

Inch or metric selection is performed by using G code.

Unit	G code	Min. input unit
Inch	G20	0.0001 inch
mm	G21	0.001 mm

Before the workpiece coordinate system setting and program starting, G20 or G21 should be specified beforehand.

The following change is corresponding with G20 or G21 code change

(1) Feedrate command F or E code.

(2) Position display.

(3) Offset value.

(4) Scale unit for MPG.

(5) Movement distance for incremental feed.

Note 1: G20 or G21 code when power-on is the same that of power-OFF.

Note 2: Inch/ metric selection can be performed by MDI mode. The selection state is changed by the G20 or G21 command.

Note 3: G20 and G21 can not be changed when the program is running.

Note 4: When the min. unit system of machine is not consistent with command unit, the max. shift error is half of the min. command increment, the error is not accumulated.

Note 5: The metric/inch input can be switched by the setting value on Setting page.

2.5.12 Feedrate Command Shift（**G98，G99**）

Feed/min. or feed/rev. is specified by the following G codes.

G code	Feedrate unit
G98	Feed/min.
G99	Feed/rev.

For the details, see Section 2.4.2.

2.5.13 Constant Surface Speed Control（G96，G97）

The following G codes specify whether the constant linear speed control is effective.

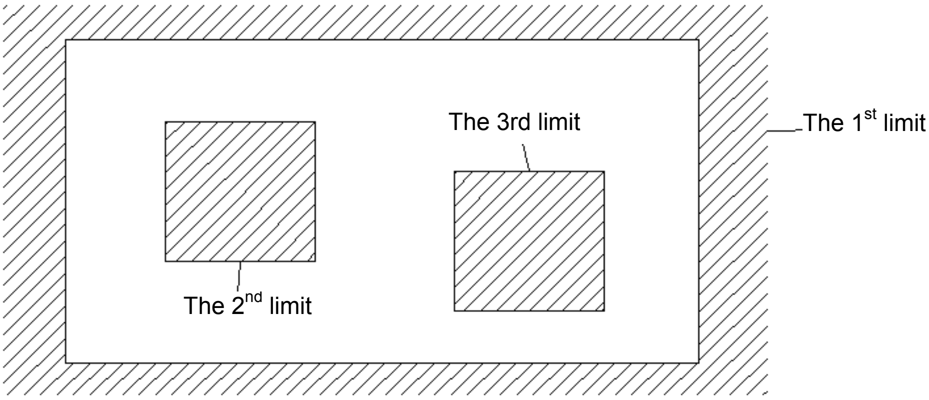
G code	Constant linear velocity control	Meaning
G96	ON	The constant linear velocity is always performed before G97 is specified.
G97	OFF	The constant linear velocity control is disabled before the G96 is specified.

For details, see Section 2.8.2.

Note: G97 (constant linear speed control off) is set when the power is turned on.

2.5.14 Stored Stroke Limit（G22，G23）

Three areas which the tool cannot enter can be specified with the following modes (tools cannot enter the shadow area).



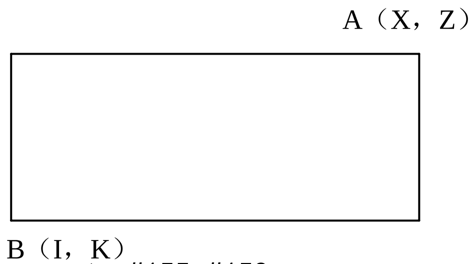
The 1st limit: Parameters set the boundary, outside area of the set limit is set to be a forbidden area. It is set by machine tool builder, once set, generally never change. Usually, this area is set with max. stroke, it is called stored stroke limit, it is also called software limit.

The 2nd limit: Set these boundaries by parameters or in MDI mode, inside area of the set limit is set to be a forbidden area. G22 command forbids the tool to enter forbidden area. G23 command permits the tool to enter the 2nd limit area and the 3rd limit area.

The 3rd limit: Set these boundaries by parameters or programs. Select either inside or outside area to be forbidden area by the parameter #86 (RWL). G22 command forbids the tool to enter the forbidden area, and G23 command permits the tool to enter the forbidden area, just like the 2nd limit area.

The 3rd limit command setting or modifying are performed by the following definition commands.

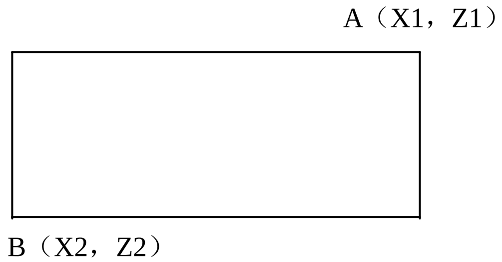
G22 X__Z__I__K;



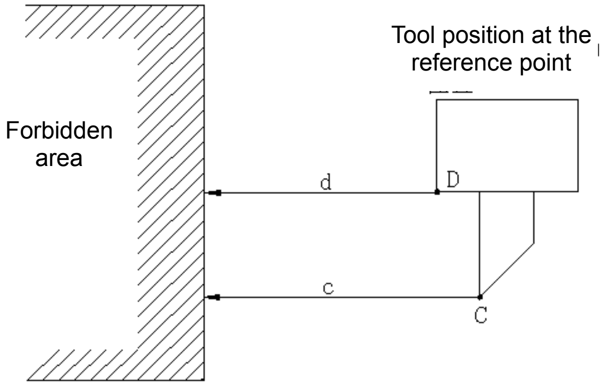
The 3rd limit can be set by parameter #155~#158.

X>I，Z>K

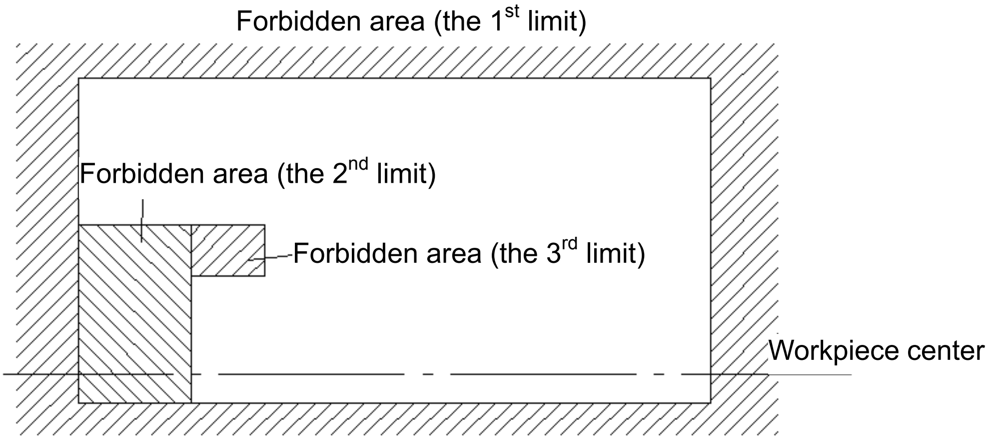
When the area is set by parameters or in MDI mode, points A and B in the figure below must be set.



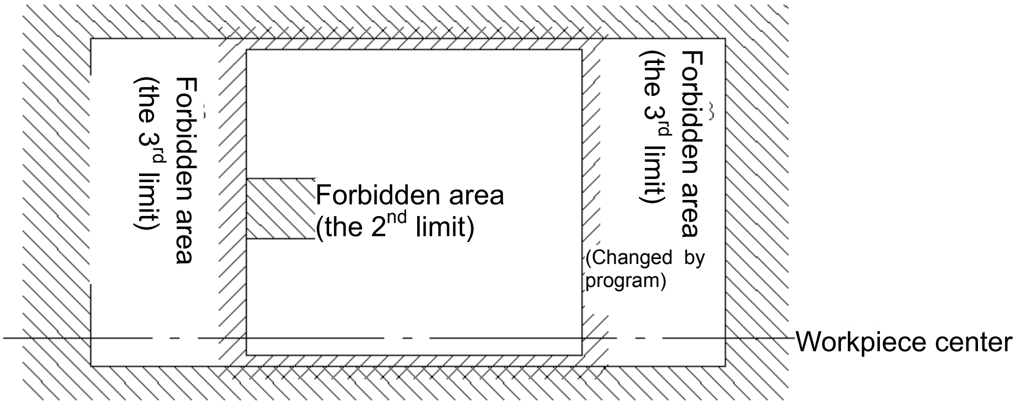
When setting the forbidden area by parameters or in MDI mode, the distance (X, Z, I and K) from reference position to point A, point B is specified with the min. movement unit (output unit) . The programmed data is then converted into the min. movement unit, is regarded as parameter setting. The data of forbidden area is determined by the position of tool checking. See the following figure:



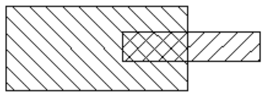
If checking from point C (tool) to enter forbidden area, the distance C should be set; and the d distance must be set, when checking from point D to enter forbidden area.



Chuck can be protected when set according to the above figure, and both the chuck and tail stock can be protected when set according to the following figure.



Area can over lapping



Note 1: After the power is turned on, each limit is enabled after the manual reference position return or automatic reference position return has been performed by G28. In order to the safety, manual can not be performed at a rapid traverse rate before (related parameter #9.0).

Note 2: If the power is turned on, the reference point return is performed, and the reference point is within the forbidden area (In the G22 mode, in the 2nd and 3rd limit), then the alarm will generate immediately. When G23 changes into G22, and if the tool is in the forbidden area, alarm occurs the next movement.

Note 3: When tool has been entered the forbidden area and it can not be retracted from the forbidden area (Note 2), pressing the ESP button. Tool is moved from the forbidden area by the G23 mode. If the setting is wrong, correcting it and the reference point return performs.

Note 4: An axis without reference point return function does not forbidden area, so, forbidden area alarm on this axis is inexistent.

The whole area becomes forbidden area when any axis does not have reference point return function, so, note that when specified with No.020 parameter.

Note 5: As for the setting of forbidden area, if two setting points are same, the forbidden area has the following definitions:

The 1st limit: No definition for forbidden area.
The 2nd limit: The whole area is forbidden area.
The 3rd limit: The whole area is forbidden area when inner area is specified. There is no forbidden area when the outer area is set.

Note 6: The set beyond the machine stroke is invalid for protection.
Note 7: Tool can be moved reversely when tool enters forbidden area and the alarm is generated.
Note 8: When the area is set, even if the dimensions of the coordinate value of two points are specified wrongly, the rectangle formed by these two points will be established this area.

Note 9: G22—; and G23; should be set in different block.
Note 10: The movement axis stops after entering forbidden area, the max. movement value 8ms stops.
Note 11: The 2nd limit is set by parameter; the 3rd limit boundary can be set by parameter or in MDI mode.
Note 12: The tool can not be retracted from forbidden area when movement axis enters the common position of forbidden area which is set by the 1st, 2nd and 3rd limits. The tool can be retracted by performing the operation in Note 3.

2.5.15　Skip Function（G31）

The linear interpolation of X (u) and Z (w) are specified by G31, like G01, if the skip function is input during the command execution, and the rest part of the block executed should be stopped to execute the next block.

G31 is a one-time completion command, only the block is enabled.

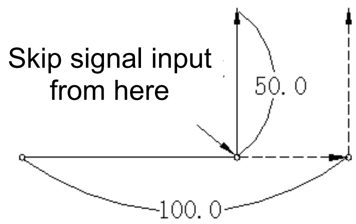
The movement after the SKIP function input is determined by in the next block is incremental or absolute.

1) When the next block is incremental command.

The next block from the breakpoint is performed with incremental movement.

For example: G31 W1000.0;

U50.0；



Actual movement

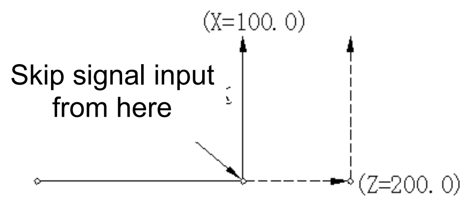
Movement without skip signal

2) When the next block is absolute command (Only one axis is specified)

Tool moves to the specified position along with the specified axis, when the skip signal is introduced, the other axes keep invariable.

For example: G31 Z200.0

X100.0；



Actual movement

Movement without skip signal

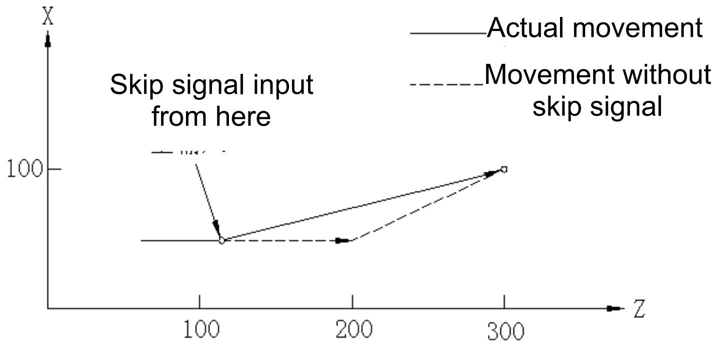
3) When the next block with absolute command is to be specified the two coordinate axes.

Tool will move to the specified position and regardless of where is the signal input.

For example: G31 Z200.0;

X100.0 Z300.0；

Chapter 2　Programming



Feedrate of G31 block is set by parameter #306 (SKPF) with the following two methods.

(1) F specifies the feedrate (F code is specified by the previous of G31 block or specified in block)

(2) Parameter No.342 sets the feedrate.

User macro program can employ the tool position coordinate value when the skip signal is ON, and it is stored into the system variables #5061 and #5062.

#5061 X coordinate value

#5062 Z coordinate value

Skip function can be employed when the movement value is indefinite, see the following:

(a) Grinding wheel standard dimension federate.

(b) Tool contacts sensor to feed tool measure.

Note 1: G31 is generated alarm No.35 when tool nose compensation R is employed.
Note 2: When G31 is commanded, the parameter sets the feedrate. When it is dry run, the feedrate is also determined by the parameter.
Note 3: When the G31 is commanded, if the feedrate is set by the parameter, and when the skip function is employed, the automatic acceleration/deceleration for adding automatic measure accuracy is not performed.

2.6 Compensation Function

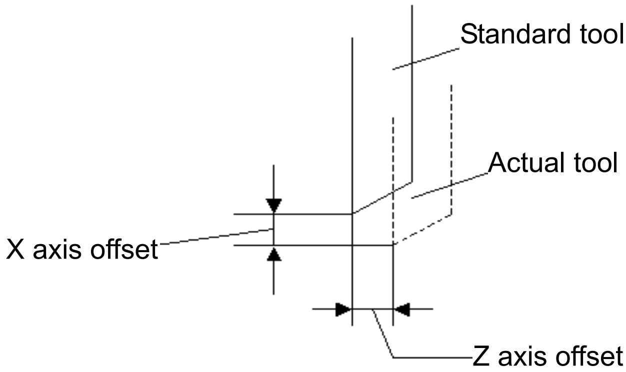
When the tool employed is different from the image tool used in programming, or when the tool nose compensation is performed by NC, this compensation function is employed. And the function includes tool offset and tool nose compensation.

2.6.1 Tool Offset

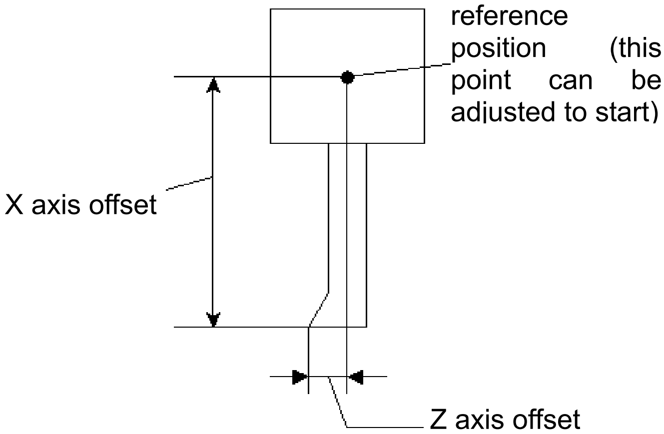
Tool offset is not specified by using G code but T code.

2.6.1.1 Reference Tool Offset

When the actual tool is different with the imaginative tool in programming (normally, it is standard tool), tool offset compensation is employed.



The standard tool nose moves along the expectant path by generally programming. In this case, tool nose is consistent with programmed starting point. However, the actual tool is hard to match the tool exactly. To compare the tool nose position setting, it is very convenient to measure the distance between standard tool and actual tool nose position. This distance is used for offset value.

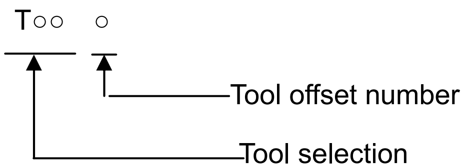


There two modes can determine the reference position: the point, which is at the head of position after the reference point return is completed, is the reference point, or some reference position is regarded as the starting point of program after reference position return is completed.

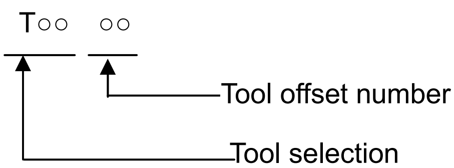
2.6.1.2 Tool Offset T Code

T codes have following meanings:

When T2+1:



When T2+2:



G10P00

The selection of T2+1 or T2+2 depends on parameter #8.3 (T2D) setting.

Note : T code selection depends on setting of machine tool builder.

2.6.1.3 Tool Selection

Tool is selected by the T codes with corresponding tool numbers.

For the corresponding relative between tool selection number and actual tool, refer to the appropriate manual issued by the machine tool builder.

2.6.1.4 Tool Offset Number

Tool offset number have two meanings:

If tool offset number is specified, the corresponding offset distance is then specified and also the offset function is executed, tool offset number 0 or 00 indicates the wear offset is cancelled.

Offset distance corresponds to offset number, and the offset memory should be set by MDI/LCD display panel (See Section 4.4.12).

There are three kinds of compensation for the appointed offset number

The compensated offset is performed by the X and Z axes which is called as tool position offset. Compensation R is regarded as tool nose compensation.

Offset number	OFX (X axis offset)	OFZ (Z axis offset)	OFR （Tool nose R compensation)	OFT (Imaginative tool nose direction)
01	0.040	0.020	0	0
02	0.060	0.030	0	0
03	0	0	0.20	0
04	.	.	.	.
05	.	.	.	.
:	.	.	.	.
:	.	.	.	.

As for the details for tool nose compensation, see 2.6.2 (The imaginative tool nose direction is also used for tool nose compensation).

When T code is selected when the offset number is neither 0 nor 00, the tool position offset is enabled.

Offset value can be specified within the following range.

mm input: 0 mm～±999.999mm

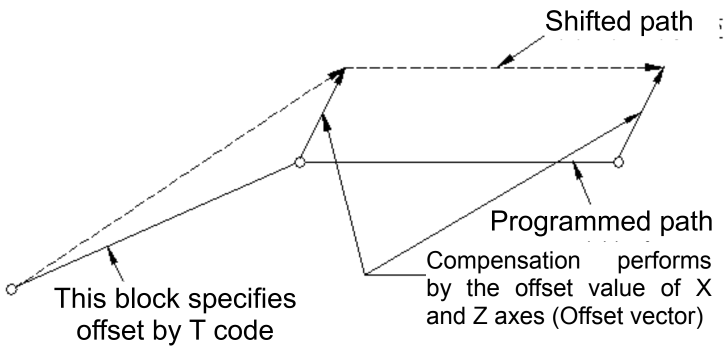
inch input: 0 inch～±99.9999inch

Note: When metric input is 1mm, the input unit of offset value is 0.001mm.

2.6.1.5 Offset

Tool path shifts programmed path X and Z values, and it corresponds to the offset value specified

by the T code, acceleration or deceleration for the end position of each block is performed.

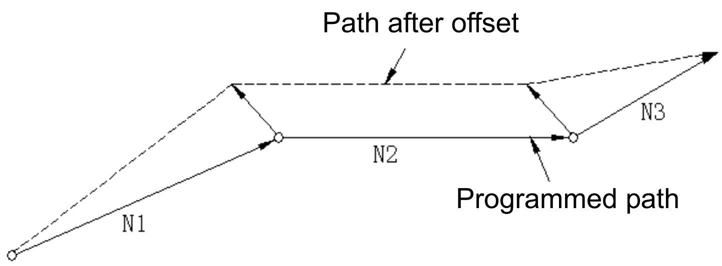


2.6.1.6 Offset Vector

In the above figure, the vector with X or Z offset value is called as offset vector, the compensation of X or Z offset value is same as offset vector.

2.6.1.7 Offset Cancel

When T code offset No.0 or 00 is applied, the wearing offset is cancelled, and the offset vector becomes 0 at the end of block of erasing wearing offset amount.



N1 U50.0 W100.0 T0202;
N2 W100.0;
N3 U50.0 W50.0 T0200;

(The offset amount is separately entered into the OFX and OFZ of No.02 offset memory.)

When the power is turned on just now and the reset key or the reset signal on MDI unit is controlled on MDI/LCD display panel, the offset amount is cancelled or the offset number becomes 0 or 00.

Set parameter #8.2 (TOC) to select whether to cancel the offset amount when the reset key is pressed or signal is input by resetting.

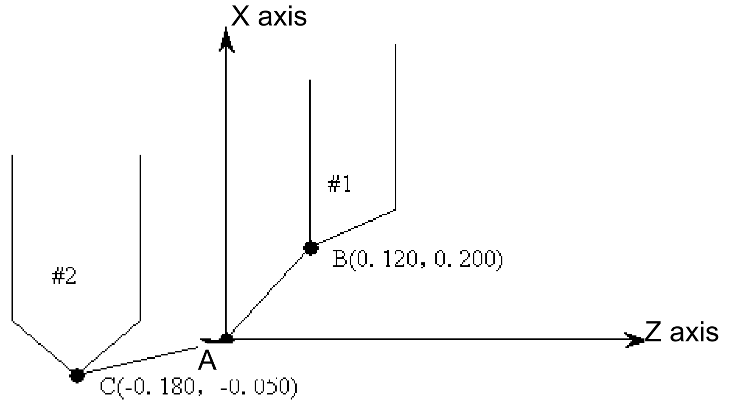
Note: When the reference position return is performed by manual or G28 or G30, the offset vector of the axis has reached the reference point is cancelled temporarily, and it is restored in the next block, but only when the next one is T code and the parameter #8.4 (TLCC) is set to 1, the offset vector can be restored.

2.6.1.8 Program Example

Tool nose coordinate value (Z, X) tool number

Tool number 1 B (0.120, 0.200) 01

Tool number 2 C (-0.180, 0.050) 02

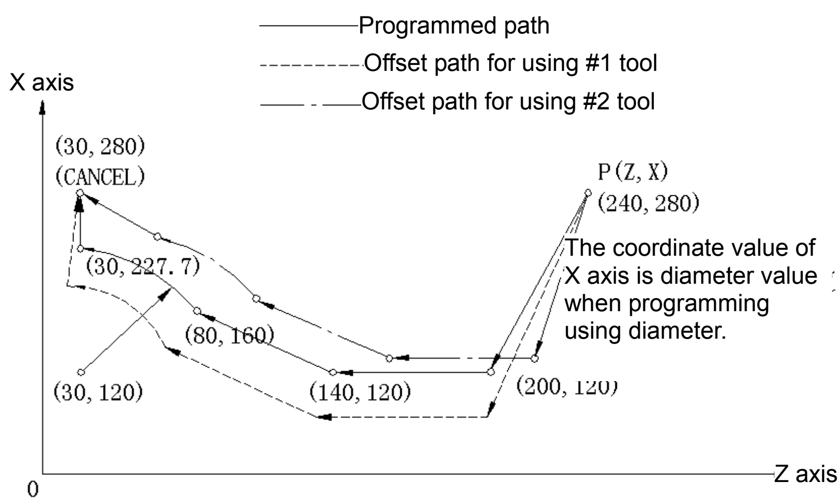


A: Tool nose setting on program.

B: Actual tool nose (#1)

C: Actual tool nose (#2)

	Tool number	
	01	02
Z	-0.200	+0.050
X	-0.120	+0.180
R	0	0



Program example 1:

```
G50 X280.0 Z240.0;
G00 X120.0 Z200.0T11;
G01 Z140.0 F30;
    X160.0 Z80.0;
G03 X227.7 Z30.0I-20.0 K-50.0;
G00 X280.0 T10;
```

Tool nose path of no.1 tool is consistent with the programmed path in this program.

For example 2:

When the following diversification is applied in program example 1, the No.2 tool nose path is consistent with programmed path. T11→T22; T10→T20.

2.6.1.9　Using T Code Separately

When only a T code is specified in a block without movement command, the tool moves the offset value at rapid traverse speed in G00 mode, and at cutting feedrate in other modes.

Wear offset movement cancellation is executed when T code of No.0 or 00 is separately specified.

Note 1: G50X (x) Z (z) T_;
Tool not moves.

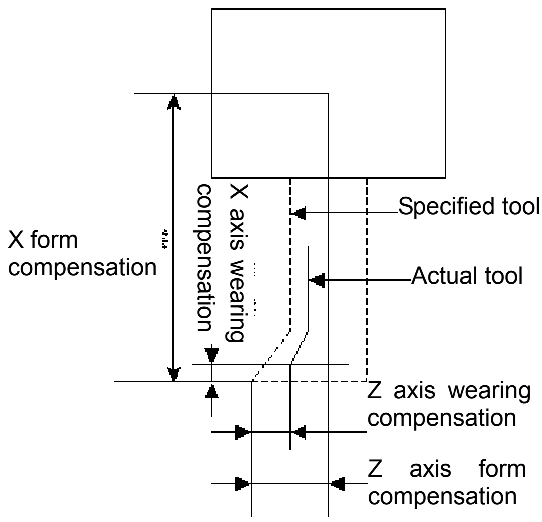
Tool position coordinate value (X, Z) can be set in the coordinate system, and it is gained by subtracting the offset value whose number is specified by corresponding T code.

Note 2: When the offset amount whose number is employing is input in MDI or Auto mode, it can be used only when the offset number of T code is re-specified, however, the new offset value can be valid in next block by setting the parameter #8.4 (TLCC).

Note 3: Block G04T_; tool change is performed when parameter setting TLCC is equal to 1, tool offset of next block with movement command is enabled.
Block: G04T_; tool offset is performed when parameter setting TLCC is equal to 0, but, tool offset on next block is disabled.

2.6.2　Tool Form Compensation and Tool Wearing Compensation

Tool position offset can be divided into two forms: the tool form compensation for tool form or tool install position and the tool wearing compensation for tool nose wearing.



2.6.2.1　T Code of Tool Offset

The following T codes are available

T○○ ○○

Shape compensation number or wear compensation number

Tool selection

Note 1: Tool shape compensation number and tool selection can be employed by setting the parameter.

Note 2: The tool offset is 32 pairs.

2.6.2.2 Tool Offset Number

Specifying tool offset number means that the corresponding offset amount and offset start are selected.

The wearing offset amount is 0 when tool offset number is 0, the wearing offset is erased.

(Normally, when the next tool is selected, the shape offset is not erased, and the D-value of coordinate is moved only.)

The offset amount corresponding offset number is set beforehand in offset memory by using MDI unit or G10 command.

The compensation amount should be corresponded with the specified offset number, it includes X and Z values and tool nose values. The offset with X and Z axes compensations are called tool position offset. The compensation of tool nose is called tool nose compensation.

Shape offset

Shape offset number	**OFGX (X axis shape offset amount)**	**OFGZ (Z axis shape offset amount)**	**OFGR (Tool nose R shape offset amount)**	**OFT (Image tool nose direction)**
G01	10.040	50.020	0	0
G02	20.060	20.030	0	0
G03	0	0	0.20	0
G04	0	0	0	0
.	.	.	.	.
.	.	.	.	.

Wearing offset

Wearing offset number	**OFWX (X axis wearing offset amount)**	**OFWZ (Z axis wearing offset amount)**	**OFWR (Tool nose R wearing compensation amount)**	**OFT (Image tool nose direction)**
W01	0.040	0.020	0	0
W02	0.060	0.030	0	0
W03	0	0	0.20	0
W04	0	0	0	0
W05	.	.	.	.
.	.	.	.	.

Image tool nose direction is universal for tool form and tool wearing offset. Both of them are

specified.

As for tool nose radius compensation, see Section 2.6.3 (Image tool nose direction is also employed for tool nose radius compensation)

Tool position offset is enabled when the T code is selected and the offset number is not 0.

Tool wearing offset is cancelled when T code is selected and the offset number is 0.

The following offset amounts are set as:

Metric input: From 0 to ±999.999mm

Inch input: From 0 to 99.9999inch

X axis offset amount can be specified with diameter or radius by using parameter (ORG), and the wearing offset can be limited a max. value (WOMAX) and an incremental offset input value (WIMAX) by parameter.(No.388 No.358).

Note: For the incremental input, see Section 4.4.12.2.

2.6.2.3 Offset

When wearing offset is executed, the programmed path is offset according to X and Z offset value, and it is gained by decelerating or accelerating the offset value corresponding to T code to the end of block.

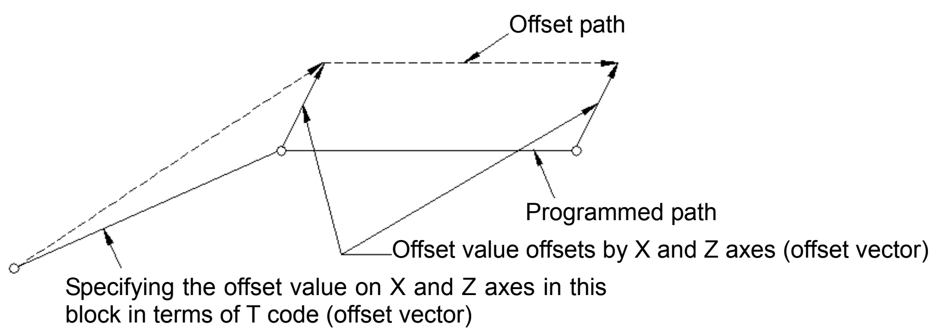


Fig.6.2.3 (a) Shape offset

Coordinate system moves the offset value of X and Z axes when it is shape offset, that is: the offset value is increased or decreased in current position.

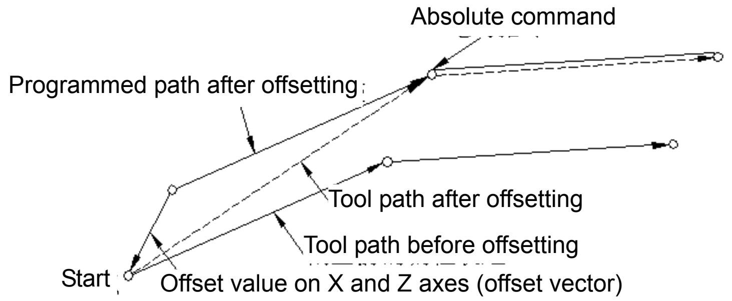


Fig. 6.2.3 (b) Shape offset

2.6.2.4　Offset Vector

A vector is composed of X and Z axes offset values, see the figure 6.2.3 (a), this is called the offset vector. The compensation performed by X and Z offset values is called the offset vector compensation.

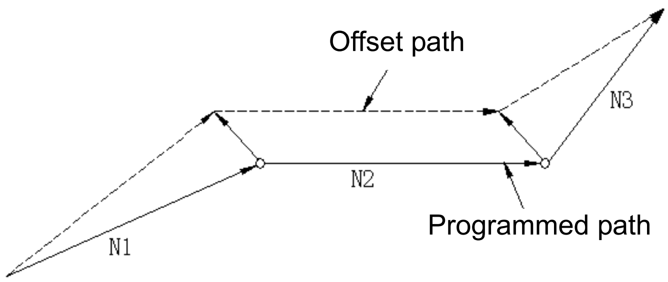
2.6.2.5　Offset Cancel

When the T code of offset number 0 is applied, the offset is erased at the end of the block, the offset vector is 0.

N1 U50.0 W100.0 T0202;　Wearing offset

N2　W100.0；

N3　U50.0 W50.0 T0200；

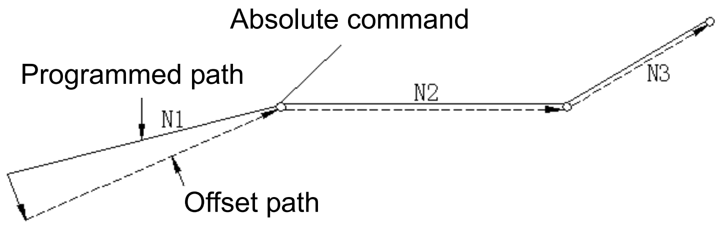


(The offset value is set in the OFWX and OFWZ of no.2 wearing offset memory)

N1 X50.0 Z100.0 T0202;　　Shape offset

N2　Z200.0；

N3　X100.0 Z250.0 T0200；



(The offset value is set in the OFGX and OFGZ of No.2 shape offset memory)

Note 1: When shape offset is generally used with tool selection, the shape offset of tool selection 0 is canceled.

Note 2: When shape offset is generally used with wearing offset number, it is set by parameter, and the shape offset of offset number 0 can be erased.

2.6.3　Tool Nose Compensation（G40～G42）

When the tool nose arc, the cutting tapper and arc are applied, and only the tool offset function is applied, it is not enough for the compensation required in forming the precision part, so tool nose compensation function is used to automatically compensate the above error.

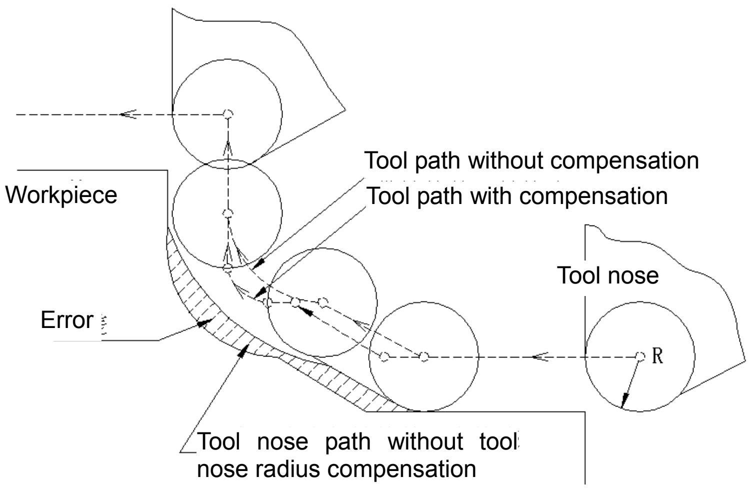


Fig 6.3 Tool path with tool nose radius compensation

Command format：$\left\{\begin{matrix}\text{G40}\\ \text{G00}\\ \text{G41}\\ \text{G01}\\ \text{G42}\end{matrix}\right\}$ X_ Z_ T_；

Command function: compensate the tool nose radius of the machined tool to improve its machining precise.

Command explanation:

G40：cancel tool nose radius compensation.

G41：specify left tool compensation（rear tool post system）.

G42：specify right tool compensation（rear tool post system）.

G00/G01：movement command.

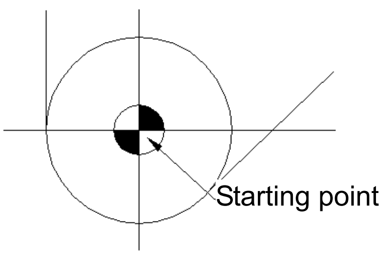
X_/Z_：movement command coordinate value.

T_ ：tool nose direction.

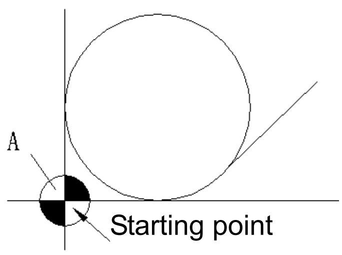
2.6.3.1 Imaginary Tool Nose

No tool nose exists in the following position A. It is necessary for the imaginary tool nose, because the starting point or base position is difficult to coincide with the tool nose center.

When tool is set to the starting point, the position relation is shown below.



Tool center is consistent with starting point



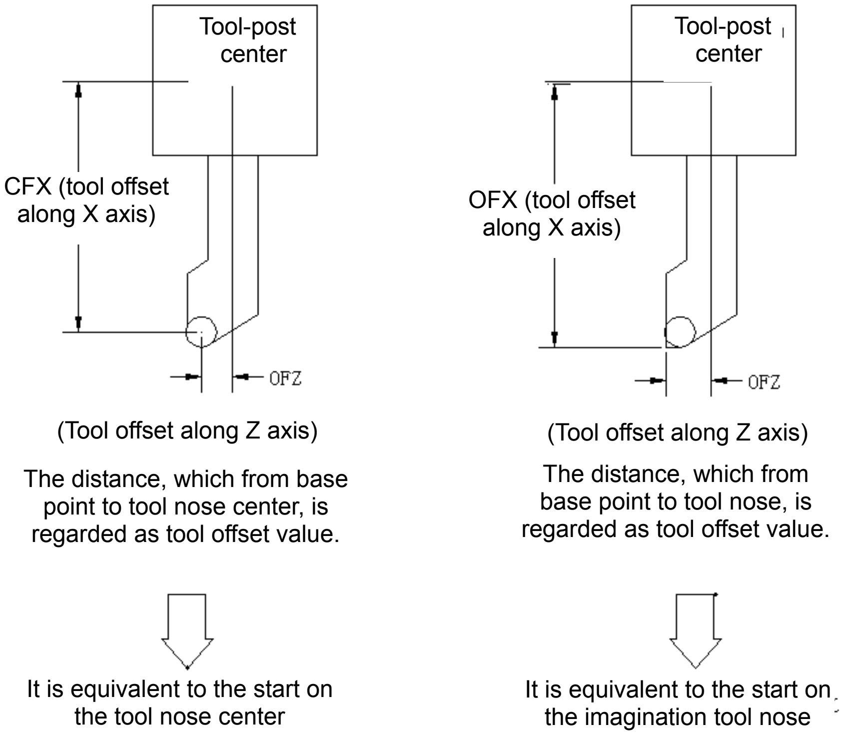
Imagination tool nose center is consistent with starting point

Note 1: Tool nose position can be not considered in programming when the imaginary tool nose is employed.

Note 2: When machine tool reference position is performed, base point as tool post center can be at the

starting point. The distance from base point to tool nose radius center or imaginary tool nose is regarded as tool position offset value. The distance from base point to tool nose radius center is regarded as offset value which is same as the tool radius center is set on the starting point, when the distance from base point to imaginary tool nose is set to be the same as the imaginary tool nose on the base point. In order to set the offset value, normally, the distance from base point to imaginary tool nose is easier than from base point to tool nose radius center.

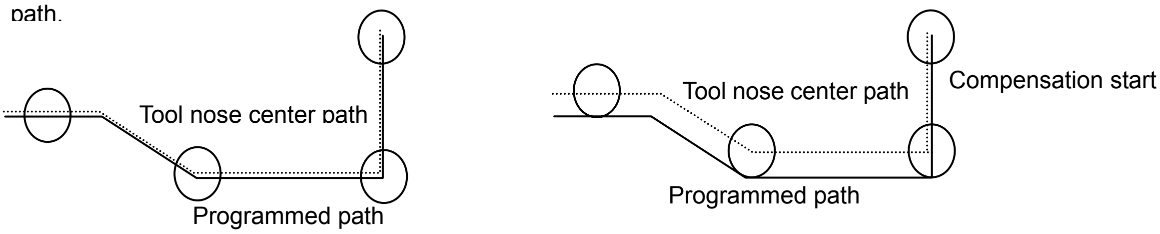
When tool-post center is put at the starting point:



I) When the tool center is machined on the starting point.

If tool nose radius compensation is not performed, the tool nose center path is same with the programmed path.

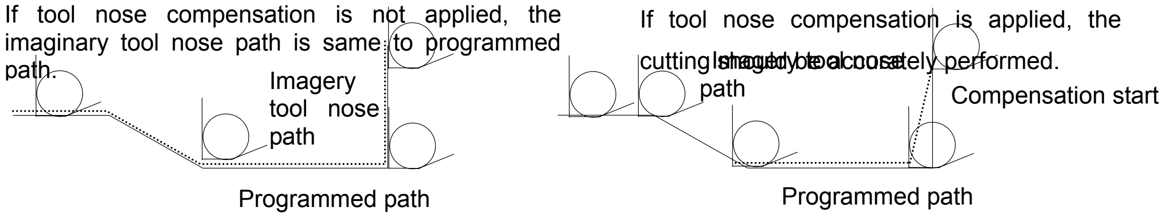
If tool nose radius compensation is applied, the cutting should be accurately performed.



Ⅱ) When imaginary tool nose is machined on the starting point:

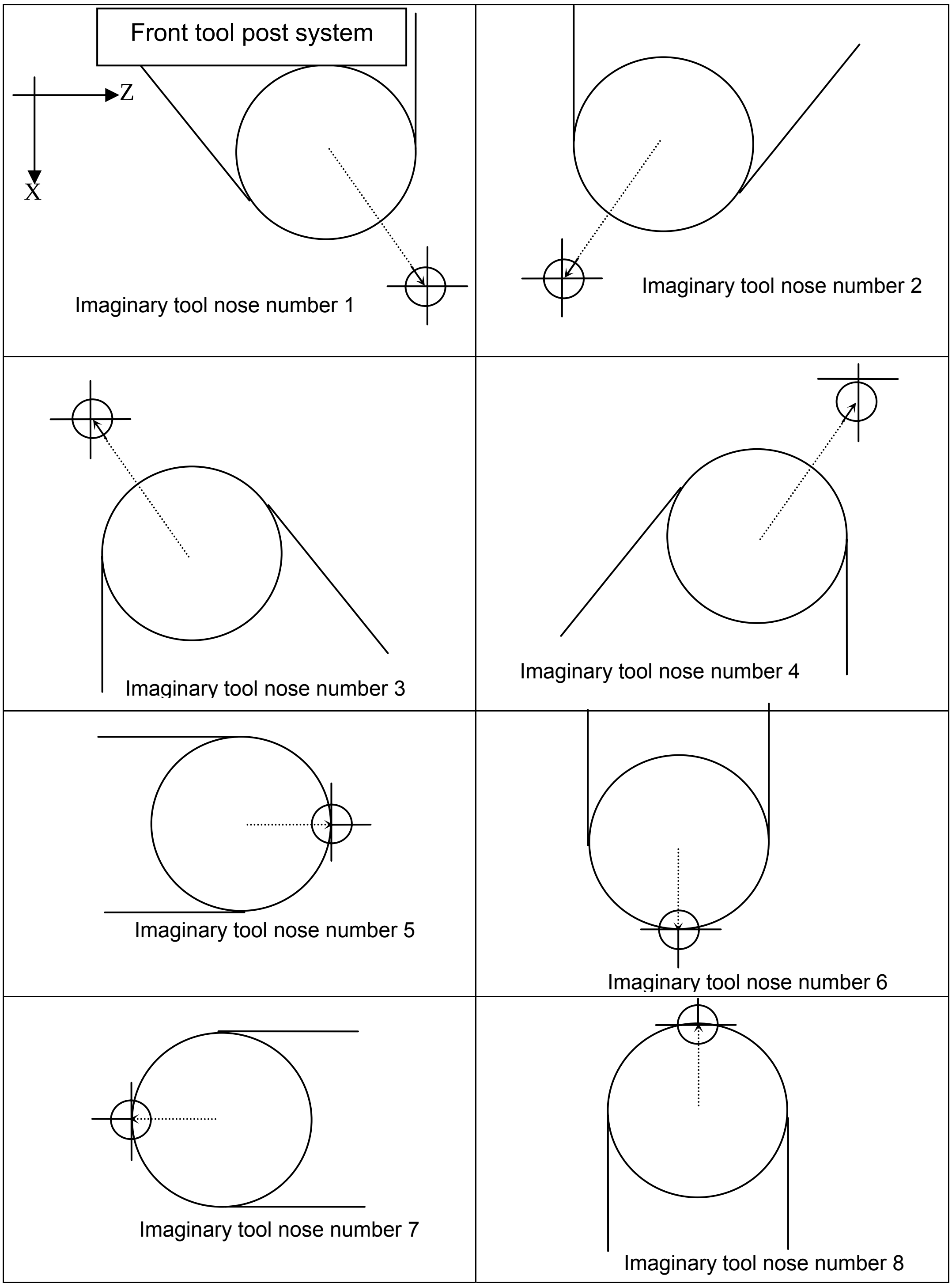
If tool nose compensation is not applied, the imaginary tool nose path is same to programmed path.

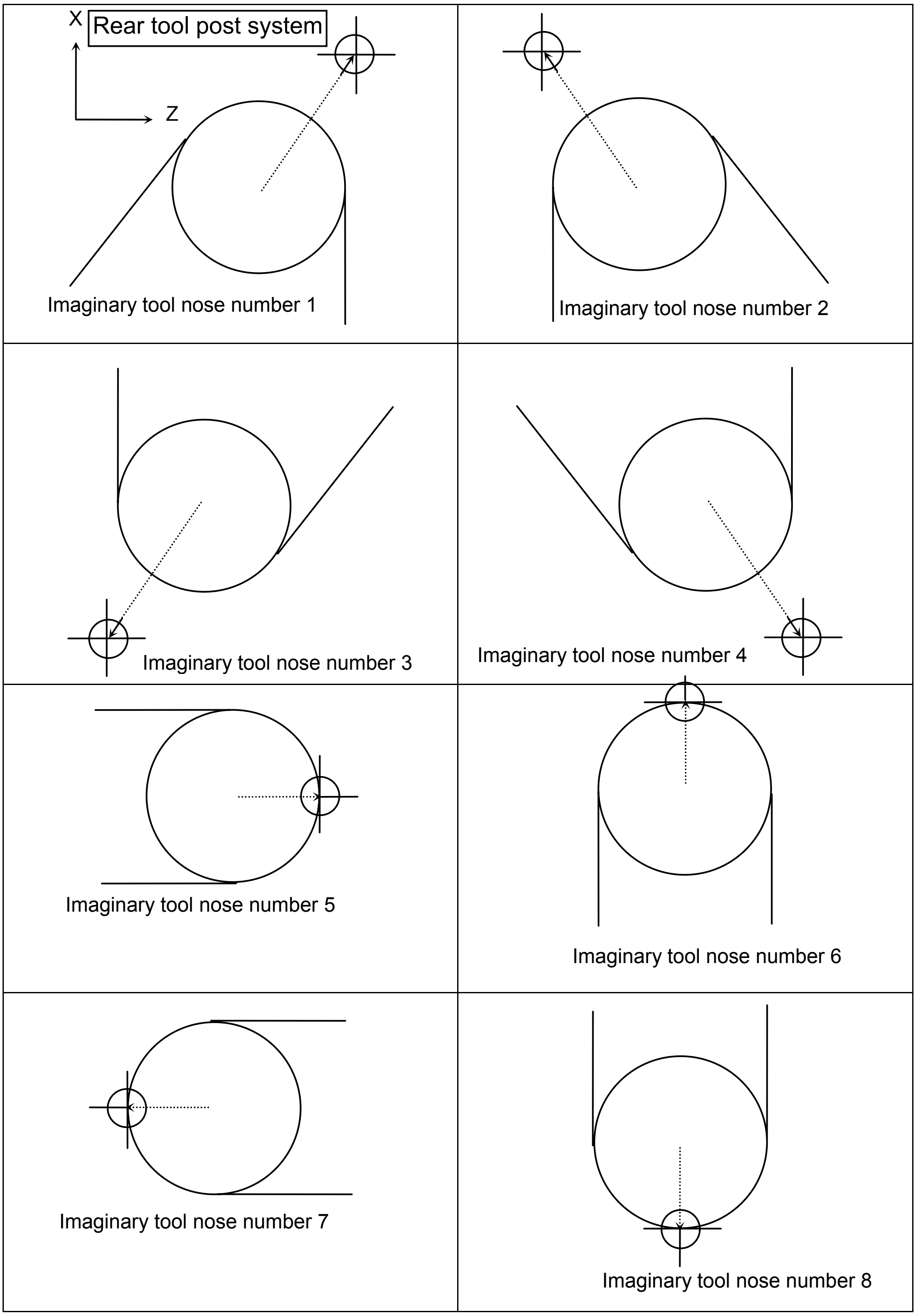
If tool nose compensation is applied, the cutting should be accurately performed.



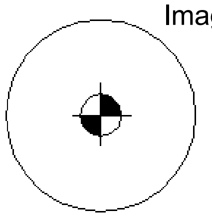
2.6.3.2 Imaginary Tool Nose Direction

The imaginary tool nose direction is determined by the tool direction which is performed from the tool nose center, so, it should be reset with the compensation value. The imaginary tool nose direction is as follows, there are 8 kinds of selections using their corresponding codes. These figures indicate the relations between tool nose and starting point, the arrows indicate these imaginary tool noses. Note that the same tool nose direction in different tool post system (front tool post and rear tool post) means different tool nose direction.

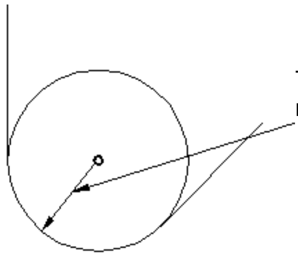




The imaginary tool nose number 0 and 9 are employed when tool nose center is consistent with starting point. Set imaginary tool nose number for according address OFT in each offset number.



2.6.3.3 Tool nose Compensation Value Setting



This value is set in MDI mode, and it corresponds to the last 2 bits numerical of 4 digits T code. (Note) the offset number specified.

T□□□□

Offset number	OFX Offset value of X axis	OFZ Offset value of Z axis	OFR Tool nose radius compensation value	OFT Imaginary tool nose direction
01	0.040	0.020	0.2	1
02	0.060	0.030	0.25	2
.	.	.	.	.
.	.	.	.	.
.	.	.	.	.
.	.	.	.	.
.	.	.	.	.
.	.	.	.	.
31	0.050	0.015	0.12	6
32	0.030	0.025	0.24	3

32 groups

Note: The offset number can be set by the least effective number of T code, it is determined by the parameter #8.3(T2D) setting. In this case, the range of offset number is 1~ 9 and the range of offset value is as follows:

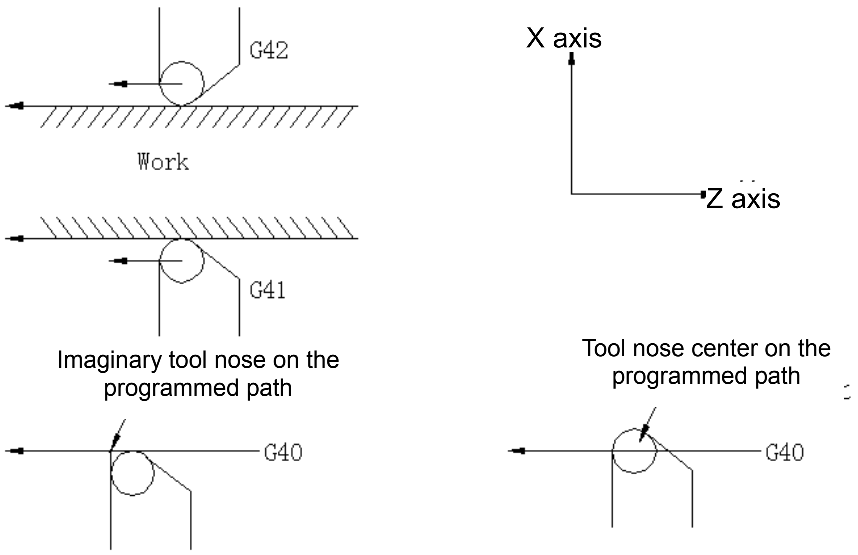
	Metric	Inch
Offset value	0 mm～±999.999mm	0 inch～±99.9999inch

Offset value corresponding offset number 00 is always 0, offset number 00 cannot set offset value.

2.6.3.4 Workpiece Position and Movement Command

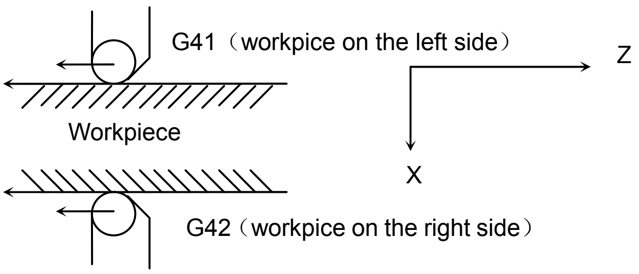
The workpiece position should be specified when compensation is performed with tool nose radius (rear tool post system).

G code	Workpiece position	Tool path
G40	(Cancellation)	Move along programmed path
G41	Left	Move at the left of programmed path
G42	Right	Move at the right of programmed path



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

Coordinate setting can change the workpiece position, which is shown below (front tool post system):



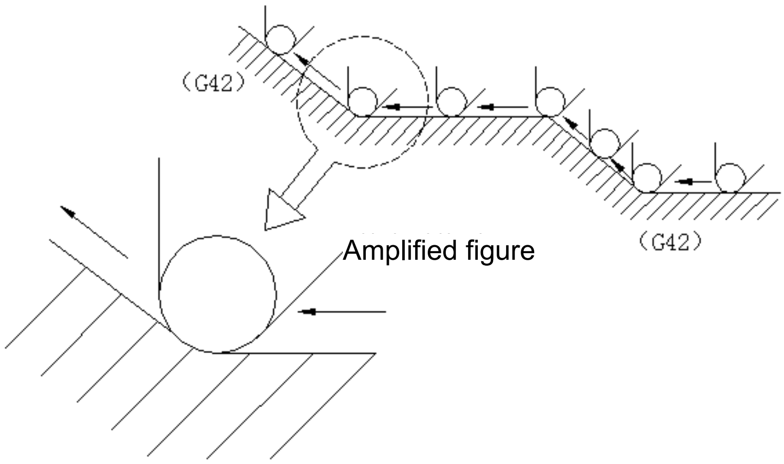
Note: 1. If the value of tool nose radius compensation is negative, the workpiece position changes.
2. G40, G41 and G42 are modal

```
G41  X··········Z··········  ;  }
     X··········Z··········  ;  }  G41 mode
G42  X··········Z··········  ;  }
     X··········Z··········  ;  }  G42 mode
G40  X··········Z··········  ;  }
     X··········Z··········  ;  }  G40 mode
```

3. G41 is not specified in G41 mode, if done, the compensation is incorrect. Similarly, G42 is not specified in G42 mode.

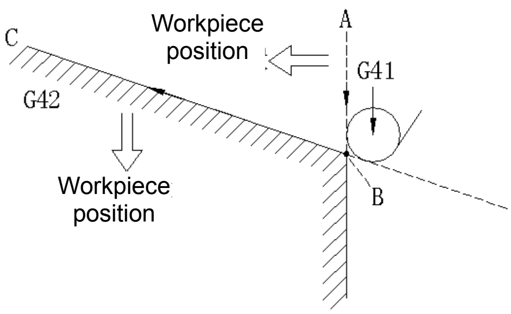
（1）When workpiece position does not change:

Tool nose keeps touch with workpiece when the tool is moving.



（2）When workpiece position changes:

The workpiece position changes relative to the tool position at the corner of programmed path, which is shown below:



Programmed path from A to B: G41

Programmed path from B to C: G42

In the above mentioned case, though the workpiece is not at the right of programmed path, it is supposed that the workpiece should exist in the movement from A to B. Because the workpice cannot change at the tool nose compensation starting at the next block, when the movement block specified from A to B is the block to start the tool nose compensation, the tool path is not same as that of the

figure.

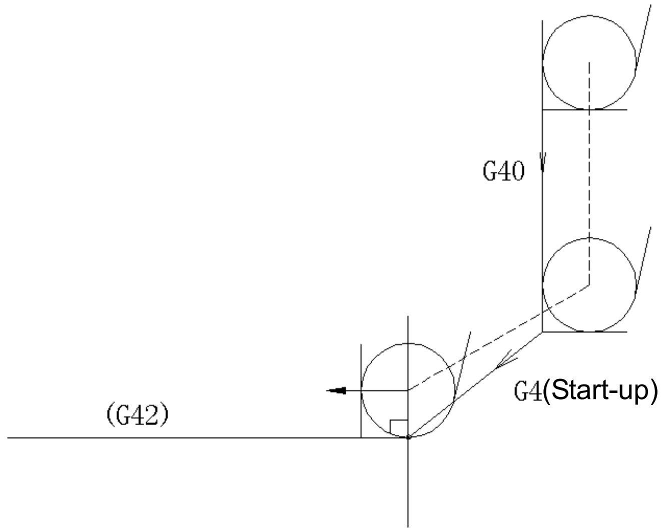
（3）Start-up tool

A block is changed from G40 to G41 or G42, is called start-up block.

G40 ————;
G41 ————; (Start-up block)

————;

The transition movement offset is performed in a start-up block; the block starts after a start-up block, tool nose center is positioned to the programmed path of the block which is vertical to the starting point.

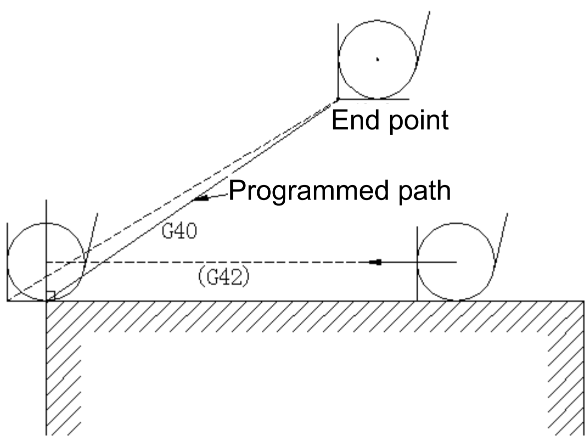


（4）Offset cancellation

A block in G40 changed from G41 or G42 is called an offset cancellation block.

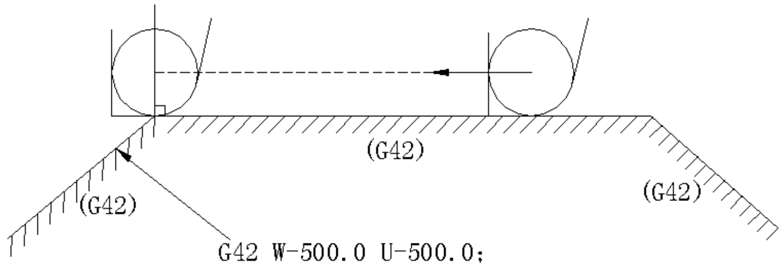
G41 __ ;
__ ;
G40 __ ; block for offset cancellation.
__ ;

At the end of the previous block before the tool offset cancellation, the tool nose center moves to a position which is vertical to the programmed path. The tool is positioned below:



(5) When G41/G42 is newly specified in mode G41/G42.

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

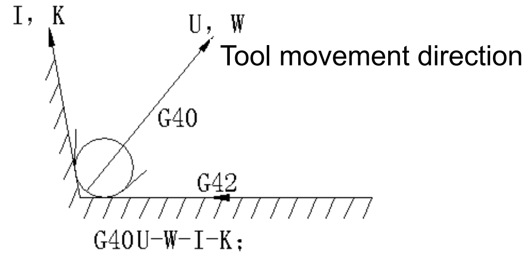


The above tool nose center position is not performed in the block in which G41/G42 is first specified.

(6) When the tool movement direction and that of workpiece are different in a block including G40 command:

If the tool retraction is performed in the specified direction, the tool nose radius compensation X (U) and Z (W) should be cancelled at the machining end of the first block in the following figure.

G40X（U）____Z（W）____I____K____；



Workpiece position address I and K must be in the same block with G40, and when address I and K are specified without G40, they are called chamfering data. When G02 and G03 are applied, they are regarded as coordinate values of arc center.

G40 X__Z__I__K__；	Tool nose radius compensation
G01X__K__； Z__I__；	Chamfering
G40 G02 X__Z__I__K__；	Arc interpolation

The workpiece position specified by address I and K is the same that of the previous block; If address I and K are specified in G40 cancellation mode, they are omitted.

G40 G01 X__Z__；

G40 G00 X__Z__I__K__； offset cancellation mode

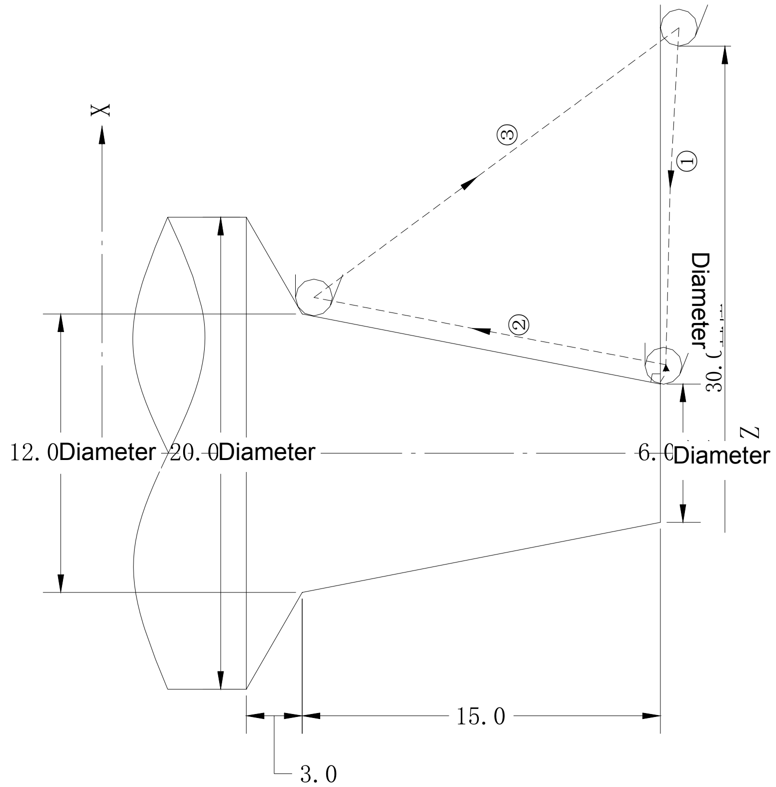
$\underbrace{\text{I__K}}_{\text{disabled}}$

G00 X__Z__I__K__；

Address I and K are regarded as chamfering data when the G40 is not specified; alarm occurs because of the incorrect format when the block is executed. I and K are specified with radius value.

Chapter 2 Programming

（7）For example



(Unit: mm)

(Diameter programs in G40 mode)

① G42 G00 X6.0 ；

② G01 X12.0 W-15.0F10 ；

③ G40 G00 X30.0 W15.0 I4.0 K-3.0 ；

2.6.3.5 Precaution for Tool Nose Radius Compensation

(1) Tool movement when two or more blocks without movement commands are continually programmed.

The block without movement commands:

① M05; M code output

② S21; S code output

③ G04 X1000; Dwell

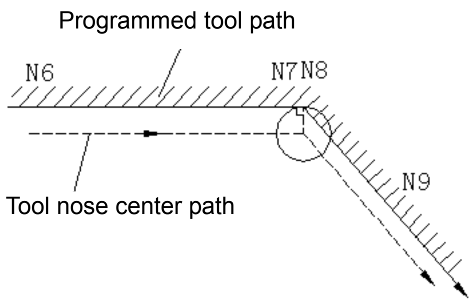
④ G01 U0; Feed distance 0

⑤ G98; G code only

⑥ G22 X10000; Machine area setting

⑦ G10 P01 X100 Z200 R50 Q2; Change offset with program

If the mentioned above two or more blocks are specified serially, the tool nose center will move to the position where the end point of the previous block vertical to the programmed path of the previous block. However, if the ①, ② mentioned above have not movement command, the tool movement is completed in the block and moves to the end point.



(G42 mode)

N6W1000.0；
N7S21；
N8M04；

N9U-1000.0W1000.0；

The overcutting generates in the mentioned above illustration.

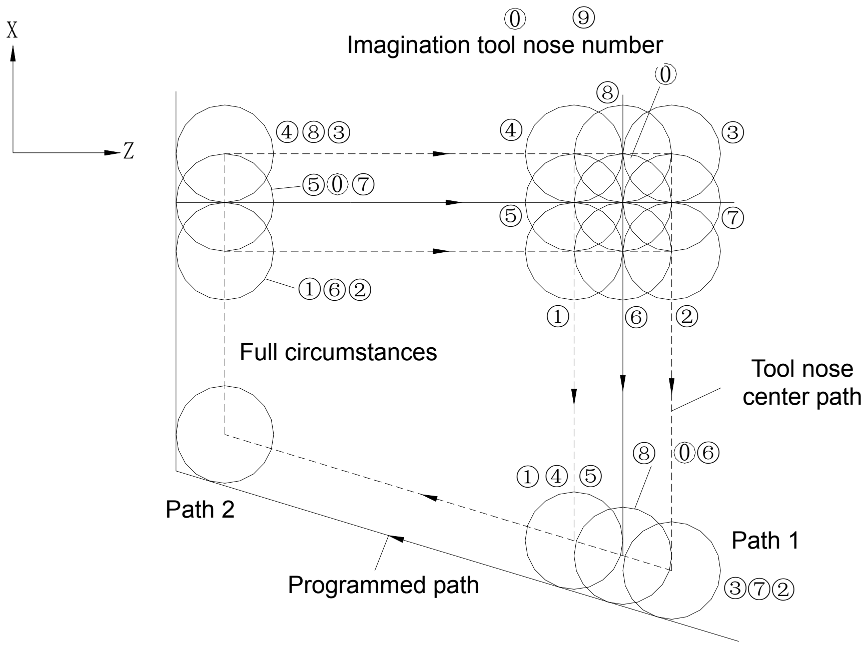
(2) Compensation is performed with G90 or G94.

The tool nose compensation is applied with G90 or G94 as follows:

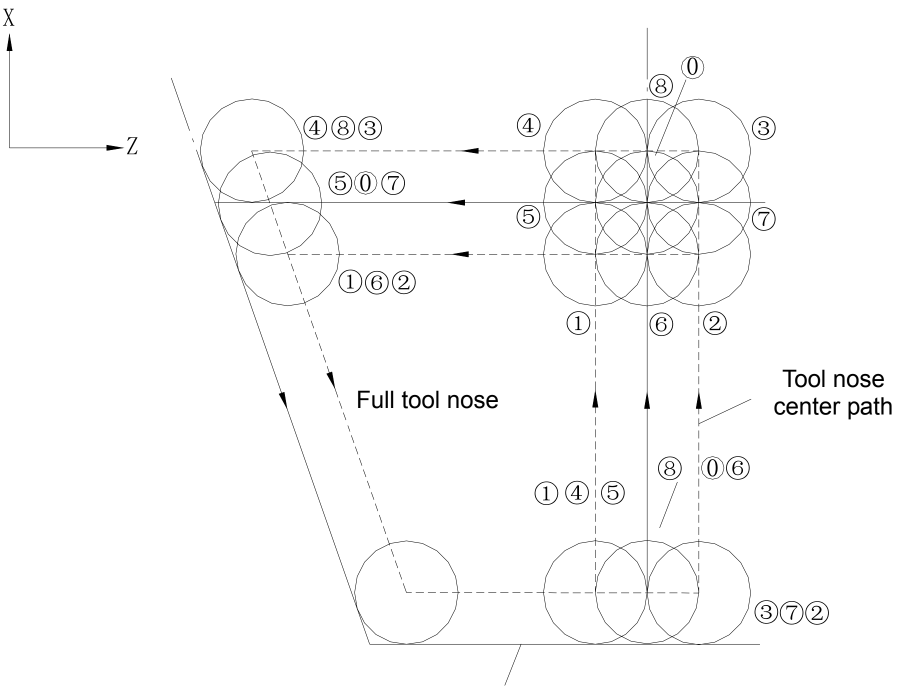
(a) Imaginary tool nose movement

Normally, the tool nose center path is parallel to the programmed path for each path in the cycle.

(I) G90 (Cutting cycle A)

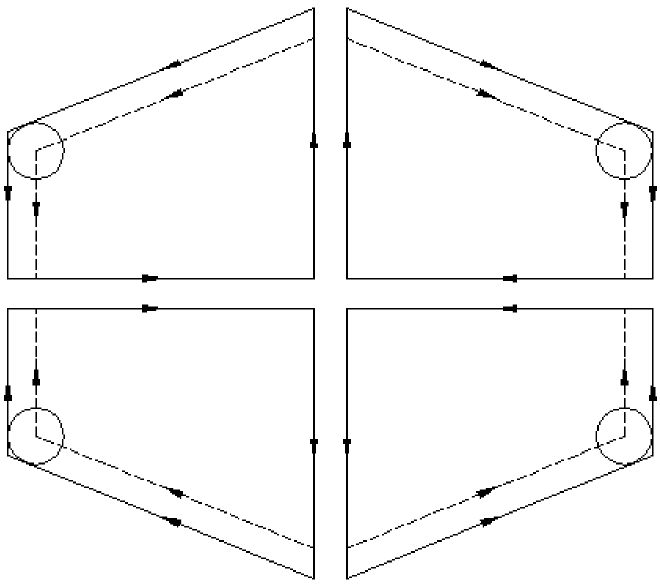


(II) G94 (Cutting cycle B)

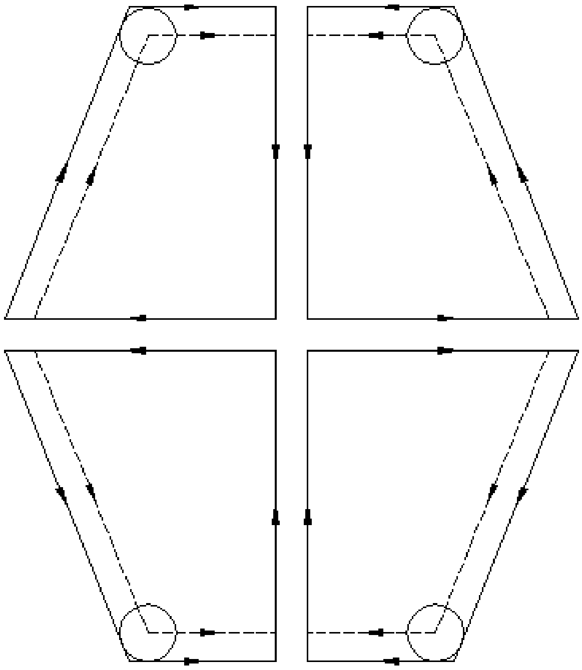


(b) If G41/G42 mode is not considered, the offset direction is shown below:

（Ⅰ）G90



（Ⅱ）G94



（3）Compensation in G71, G72 or G73

If G71 is specified with tool nose radius compensation, when machining and when tool nose center is consistent with the start, the actual tool compensation value is equal to tool nose radius compensation value to add fine-machine surplus △U and △U, but the tool nose radius compensation value in machine is erased.

$\triangle U_1 = \triangle U +$ tool nose radius compensation value (（$\triangle U \neq 0$）

$\triangle W_1 = \triangle W +$ tool nose radius compensation value（$\triangle W \neq 0$）

See the Section 2.7.2.1

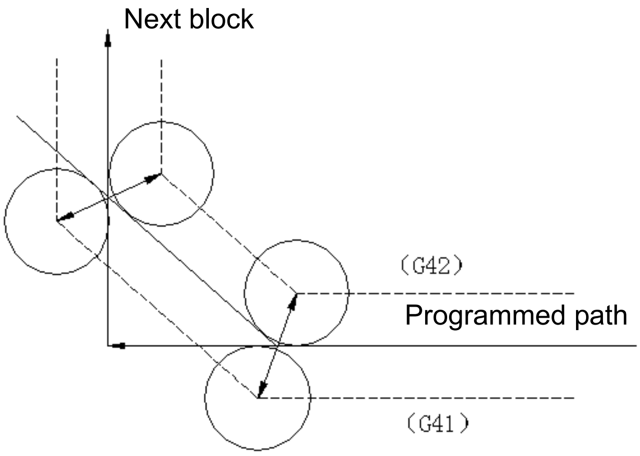
When imaginary tool nose is consistent with the start during machining, or, when the fine-machine surplus △U and △W are equal to 0, the tool nose radius compensation value is not added to the compensation.

(4) When G74, G76 or G92 is specified:

In this case, the tool nose radius compensation is not executed.

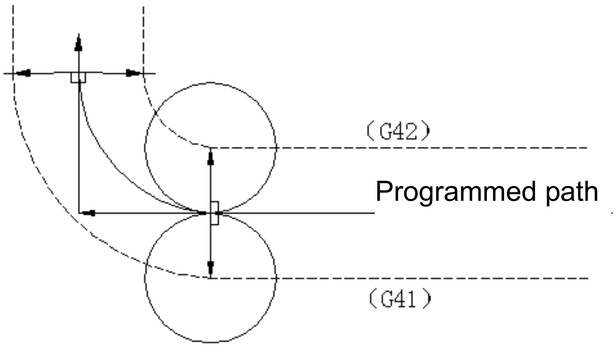
(5) When the corner executes:

The movement after compensating is as follows:



(6) When the corner arc exists:

Movement after compensating is as follows:

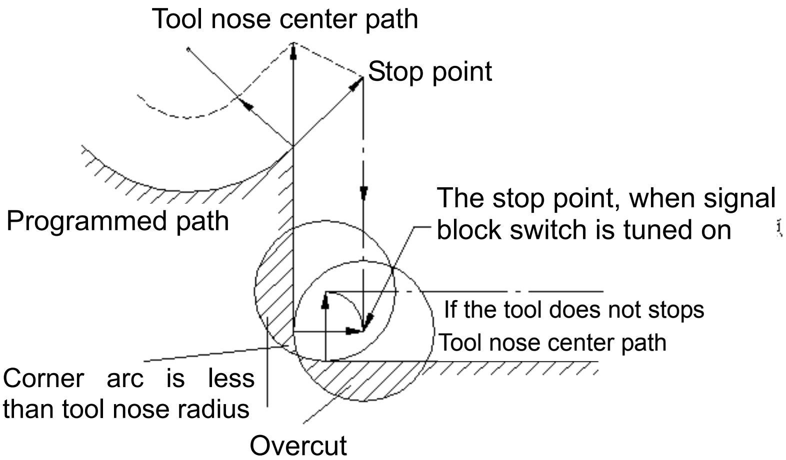


(7) Command input from MDI.

In this case, the tool nose radius compensation does not executed.

(8) When the arc inside machining is less than the tool nose radius compensation of tool nose radius:

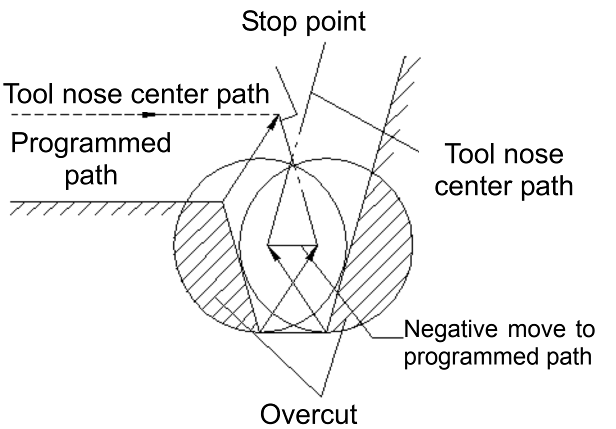
In this case, tool overcut may occur when it is in outer offset. Tool will stop to display the alarm No.41 (or after the corner moves to end) after the previous block begins. If the “signal block switch” is ON, the tool will stop at the end of previous block (The No.041 alarm occurs)



(9) The machined slot is less than the tool nose diameter.

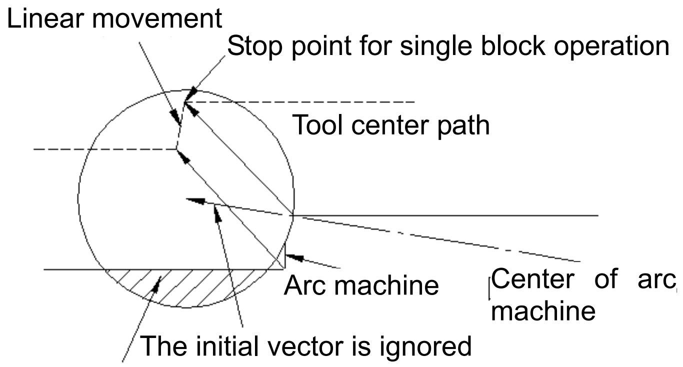
When tool nose radius compensation forms tool nose center movement path, (the reverse of program direction), the overcut is then generated.

Here, after the previous block begins (or after the corner movement is applied), the alarm (No.041) occurs and movement stops.



(10) Machined line is less than tool nose radius.

When a program of some line is less than tool nose radius, when the line is employed the arc command, normally, tool nose radius R offset in the center path is applied, and the direction is reversed to the programmed path. The initial vector is disabled at that time. It is lineal movement to the 2nd vector, and it is stopped when moving with single block. The automatic operation continues when a single block does not employ. And also it will not alarm, when commanding with line. Here, the offset is correct. (But, cutting allowance occurs)



The overcutting is not generated because the initial vector is ignored, but the movement along with arc does not execute.

2.6.4 Details for Tool nose radius Compensation

2.6.4.1 Offset Vector for Tool Nose R Center

Tool nose R center offset vector is two-dimensional and it is equal to the offset vector specified by T code, and also it is calculated in the NC system. Its direction changes along with the tool

movement corresponding to two blocks. This offset vector (it is called vector for short in the following) is generated by the tool path, which is with precision offset and calculated by program path inside the control unit (it is regarded as appropriate offset), the vector is deleted by resetting.

This vector is changed along with tool movement, it is very important to understand the vector for programming. Read and distinguish the following items and to understand how the vector generates.

2.6.4.2　G40，G41，G42

G40, G41 or G42 is used for cancelling or generating the vector, these codes are employed together with G00, G01, G02, G03 or G32 to specify the tool movement mode.

G code	Function	Workpiece position
G40	Tool nose radius compensation cancel	Both of the following are not existential
G41	Left offset applies along tool path	Right
G42	Right offset applies along tool path	Left

A kind of offset mode is specified with G41 and G42, the offset is erased when G40 is specified.

(1) Cancel mode

After power on, if the RESET key is pressed on MDI/LCD display panel or the block executes M02 or M30 and end (The system also may not enter the cancel mode for different machine tool builders). The system enters cancel mode immediately.

In the mode of cancel, the vector is set to 0; and tool nose center path is consistent with programmed path. It must be ended with cancel mode at the end of program. If the ending is applied in offset mode, the tool can not be positioned to the end point but for a position which a vector length of an end point.

（2）Starting

A block which satisfies the following conditions is executed in cancel mode, the system enters offset mode, and the control in this operation is called starting.

(I) G41 or G42 is included in block, or it enters G41/G42 mode previously.

(II) Tool nose radius compensation offset number is not 00

(III) The movement of X, Z, U or W is performed in block and the movement distance is not 0.

An arc command (G02, G03) can not be employed in start mode, if it is specified, and the No.34 alarm will be generated.

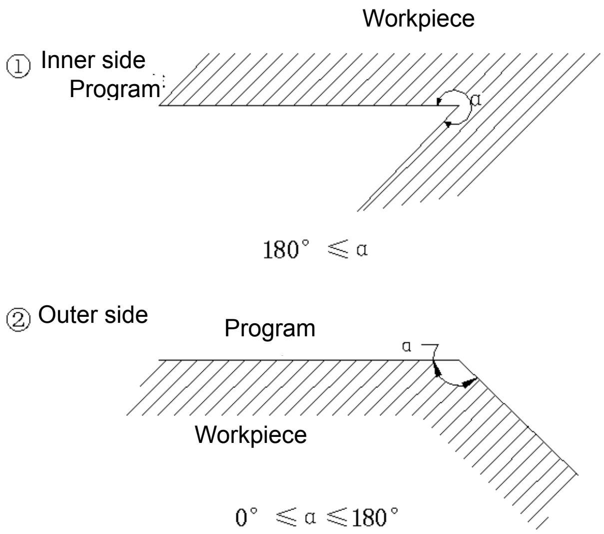
Two blocks are read in start mode, the 1st block is performed, the 2nd block enters tool nose radius compensation buffer (This data does not display).

Two blocks are read in single block mode, the machine stops when the 1st block is executed.

In the following operation, two blocks are read beforehand, a block or the next 2 blocks followed is performing in NC

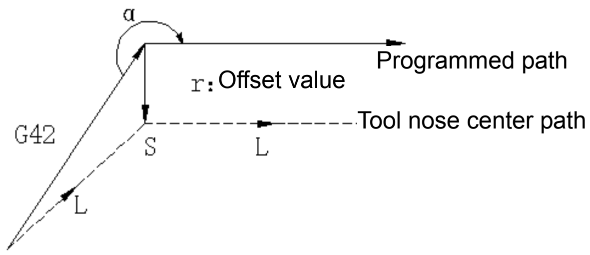
(Notice): The “Inner side” and “Outer side” explanations are as follows:

The intersection of two movement command blocks form an angle which is more than 180° viewing from workpiece side, and this is called inner side, when it is less than 180°, it is called outer side.



(a) tool movement in inner side (180°≤α)

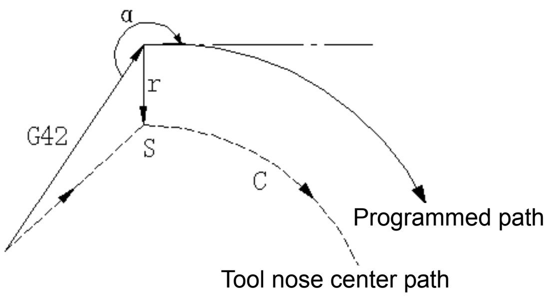
(Ⅰ) Linear→linear



S, L and C have the following meanings:

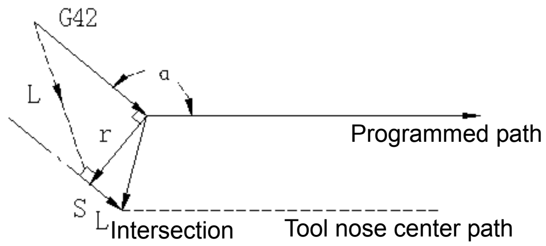
- S is single block stop point
- L is linear interpolation movement
- C is arc interpolation movement
- R is tool nose radius compensation value

(Ⅱ) Linear→arc



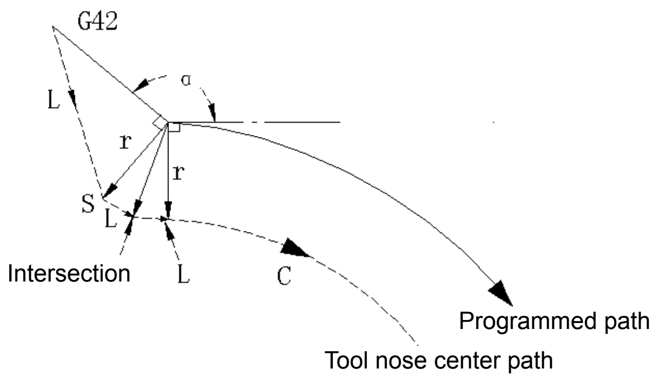
(b) Tool movement along outer side of obtuse angle (90°≤α<180°)

(Ⅰ) Linear→linear



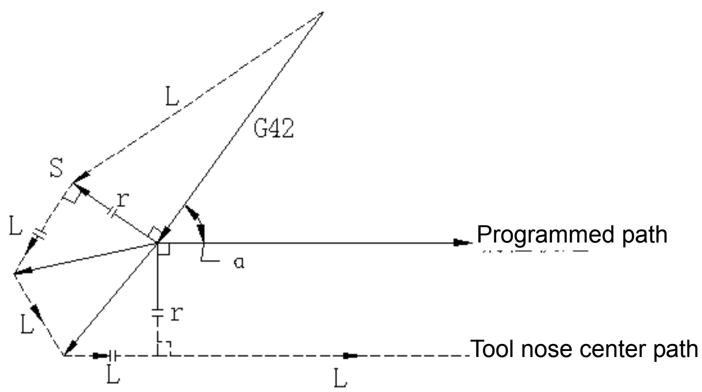
Note: the intersection is an intersection of offset paths of two block, and the programmed path is offset by r.

（Ⅱ）Linear→arc

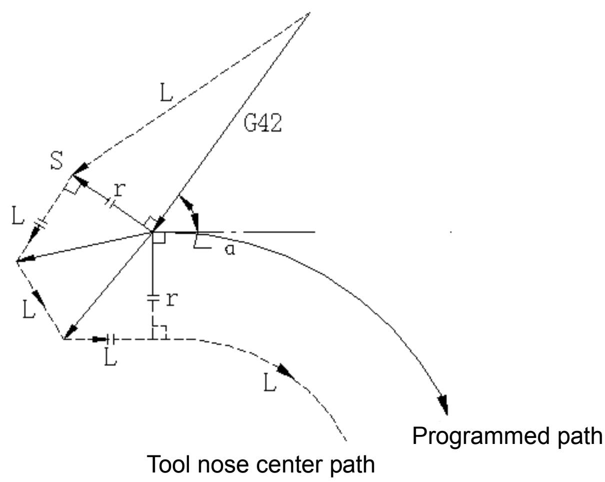


（c）Tool movement along outer side of acute angle (α＜90°）

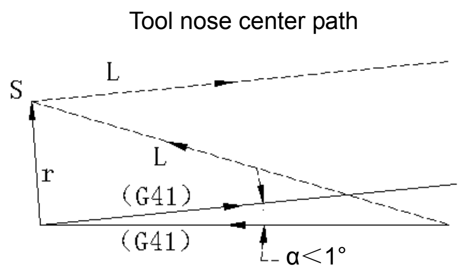
（Ⅰ）Linear→linear



（Ⅱ）Linear→arc



（d）Tool movement along outer side of acute angle（α＜1°）

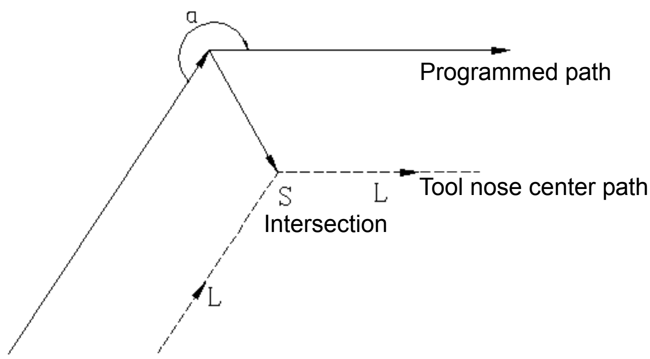


（3）Tool movement in offset mode

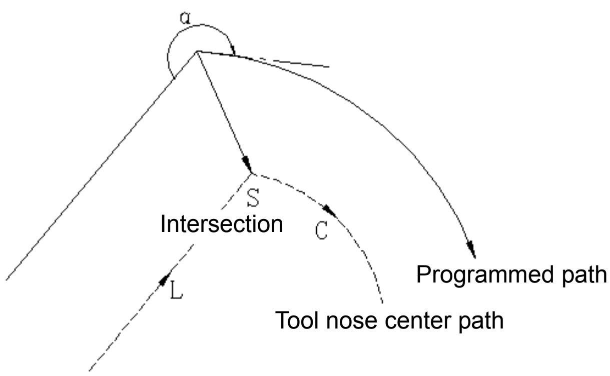
In offset mode, two or more non-movement commands are not specified, compensation is correctly executed, otherwise, overcut or short of cutting occurs.

（a）Tool movement along inner side of corner（180°≤α）

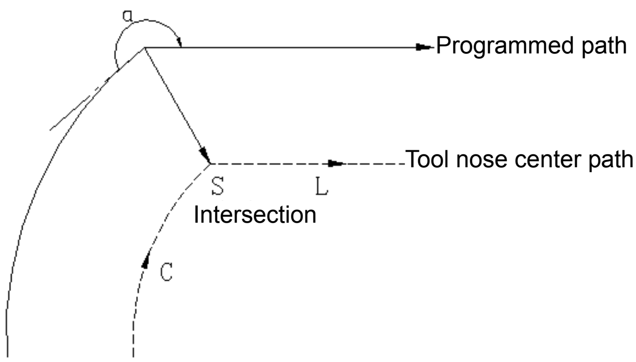
（Ⅰ）Linear→linear



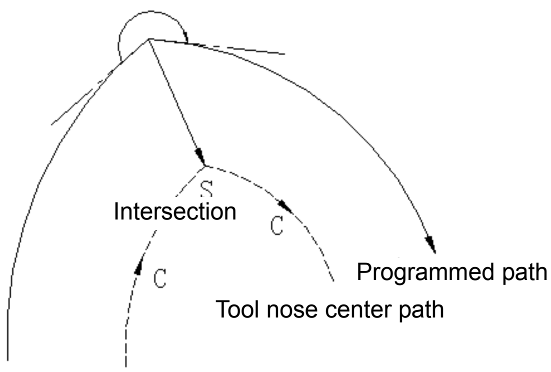
（Ⅱ）Linear→arc



（Ⅲ）Arc→linear

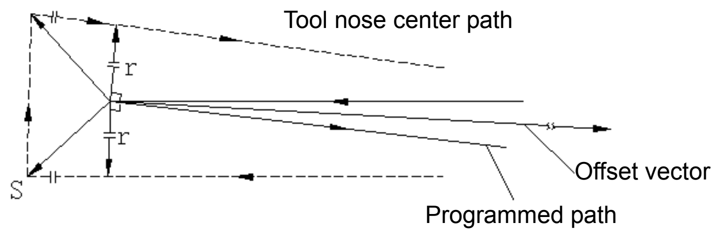


（Ⅳ）Arc→arc



（Ⅴ）Offset vector is magnified abnormally when the machining is executed at inner side of acute angle（α<1°）.

（Ⅰ）Linear→linear



The following cases do the same ways:

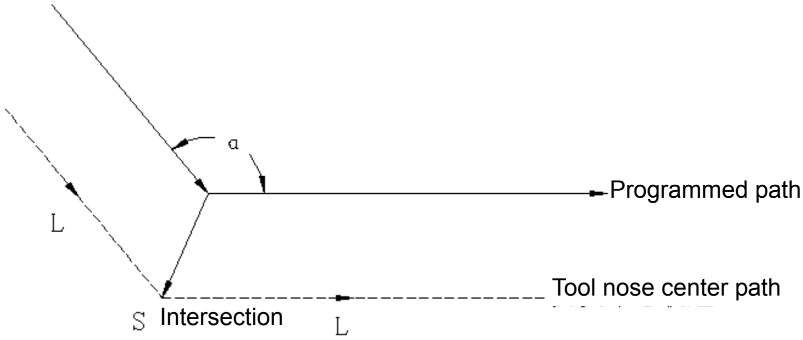
（Ⅱ）Arc→linear

（Ⅲ）Linear→arc

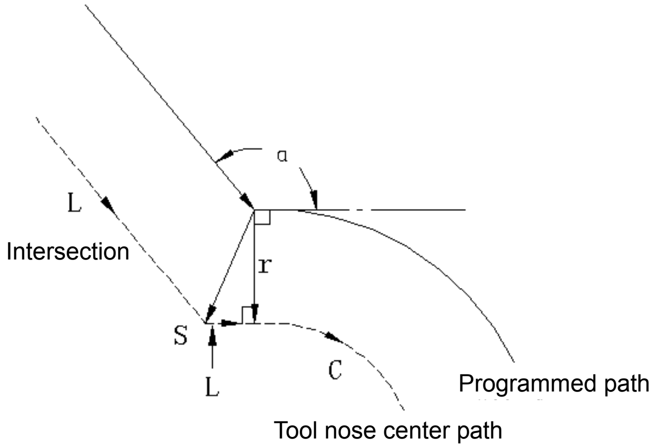
（Ⅳ）Arc→arc

（b）Tool movement along outer side of obtuse angle〔90°≤α<180°〕

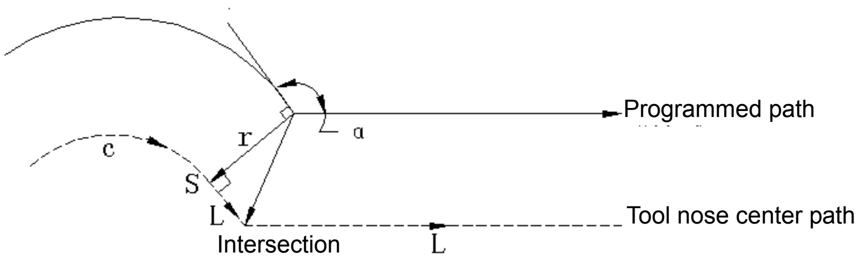
（Ⅰ）Linear→linear



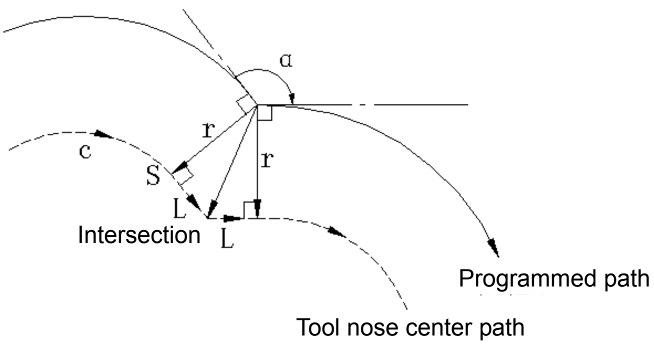
（Ⅱ）Linear→arc



（Ⅲ）Arc→linear

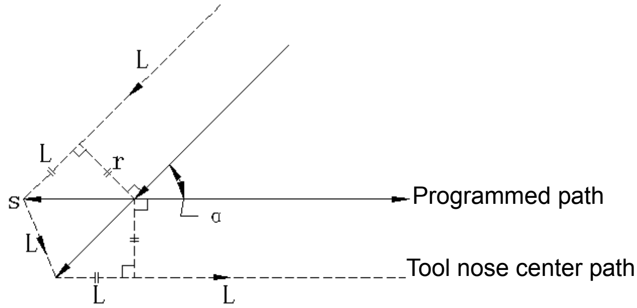


（Ⅳ）Arc→arc

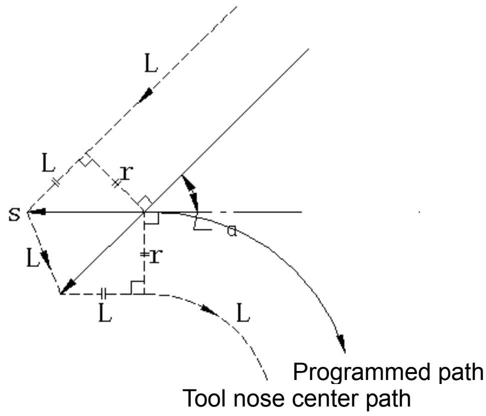


（c）Tool movement along outer side of acute angle（α＜90°）

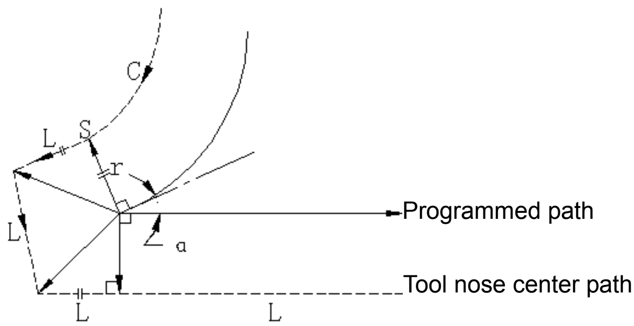
（Ⅰ）Linear→linear



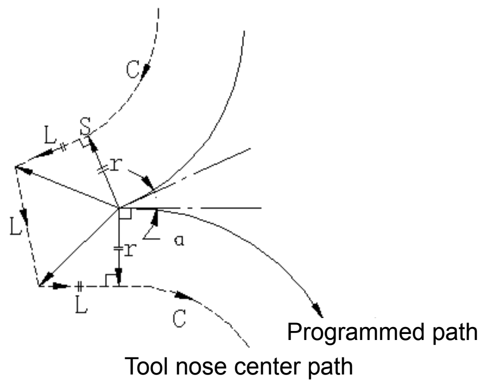
（Ⅱ）Linear→arc



（Ⅲ）Arc→linear



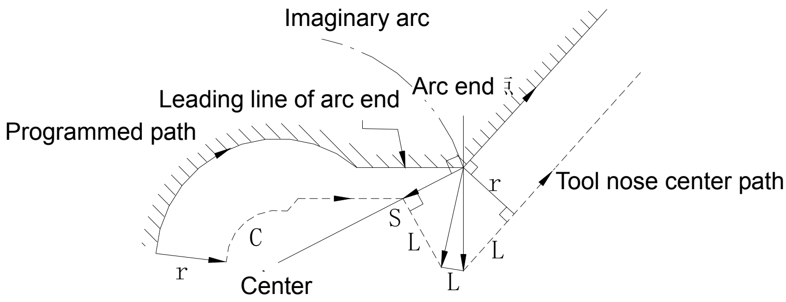
（Ⅳ）Arc→arc



（d）Exceptions

（Ⅰ）End position for the arc is not on arc, that is to make the end point of a down-lead of the arc

as the arc end to program incorrectly, see the following figure:

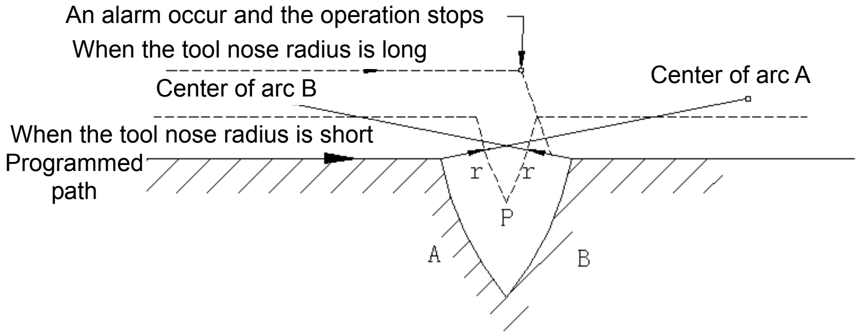


An imaginary arc is supposed in NC of above figure, and vector compensation is performed according to imaginary arc. So, tool path is different from the compensation path of the parallel extension line of arc end.

Same concept can be employed to the occasion of arc to arc.

(II) No intersection

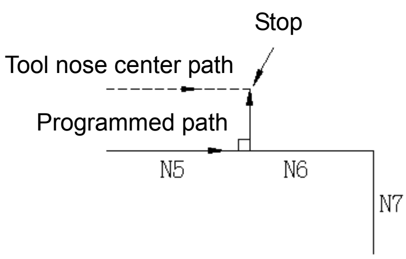
In the following figure, when the tool nose radius offset value is small, the arc offset path intersection P occurs; when the value is large, there is no intersection, and the tool stops at the end of previous block and the No.033 alarm occurs.



If the radius is small, the intersection P of offset path exists; when the radius is large, the intersection cannot be found.

（III）The arc center shares a same point with starting point or end point

When an arc is specified and its arc center shares a same point with starting point or end point, No.038 alarm occurs and the tool stops at the end of previous block.



（G41）

N5 G01 W1000；

N6 G02 W1000 I0K0；

N7 G03 U-1000 I-1000；

（4）Tool movement when offset mode is cancelled

When one of the following conditions is satisfied in offset mode and when the block is performed, the system enters cancel mode, this operation is called offset cancel.

（Ⅰ）Specified G40

（Ⅱ）The offset number 00 of tool nose radius is specified.

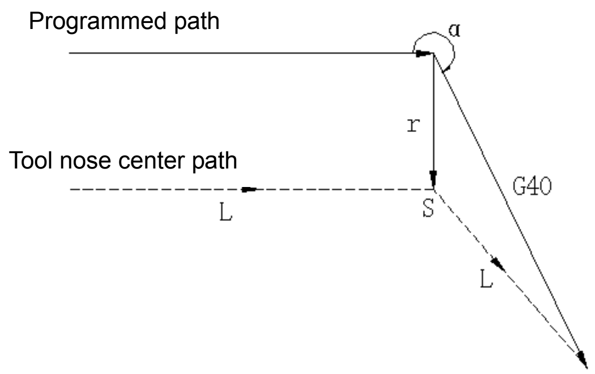
G02, G03 can not specify the offset cancel in arc commands, if it is specified the No.34 alarm will be issued.

When a block is read into and then two blocks [the program is stored into tool nose radius compensation buffer is included (this data is not to be displayed)] are performed during erasing the offset value.

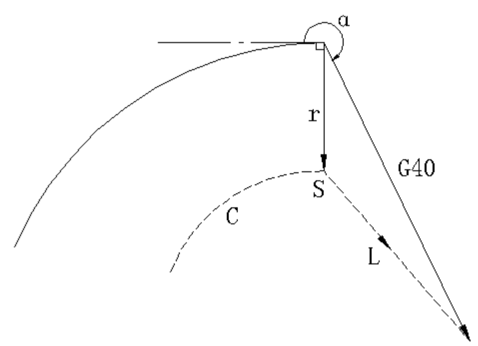
In a single block mode, one block is read into, the other is applied and stopped, and the next block is performed pressing START key, but the next block is not to be read into. The system enters cancel mode therefore, and only next block entered to the buffer is performed. The data does not read into the tool nose radius compensation buffer.

（a）Tool movement along inner side of corner（180°≤α）

（Ⅰ）Linear→linear

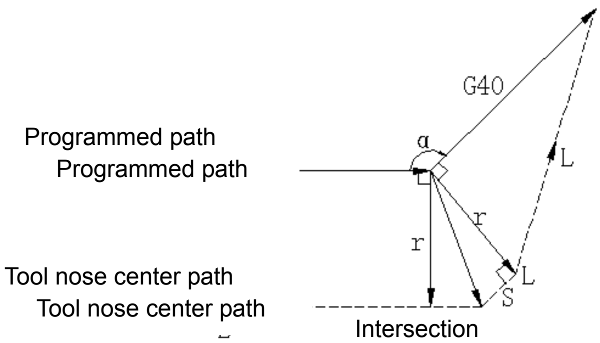


（Ⅱ）Arc→linear

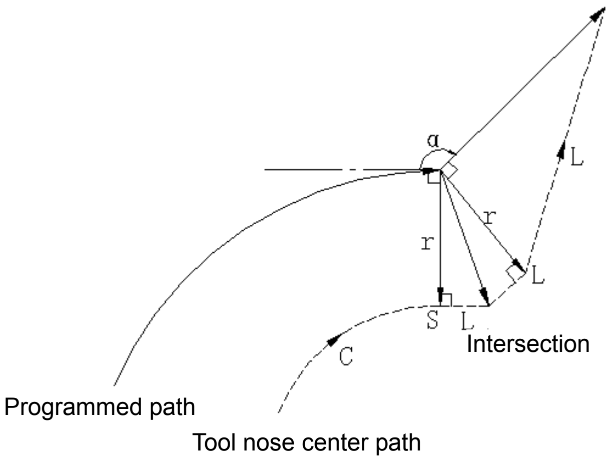


（b）Tool movement along outer side of obtuse angle（90°≤α＜180°）

(Ⅰ)Linear→linear

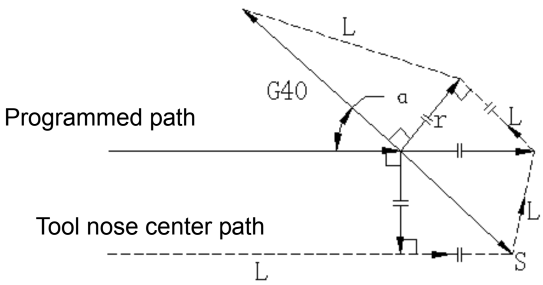


（II）Arc→linear

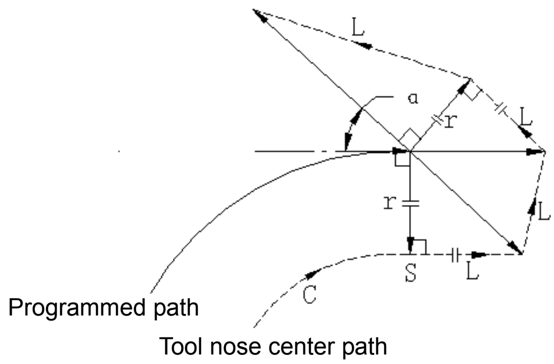


（c）Tool movement along outer side of acute angle（α＜90°）

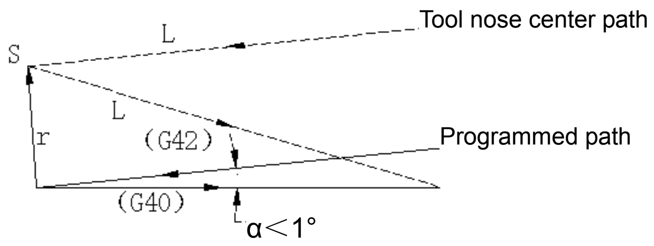
（I）Linear→linear



（II）Arc→linear



（d）Tool movement linear→linear along acute angle(α＜1°) .



(5) Offset direction changes in offset mode

The offset direction is determined by too nose radius compensation G codes (G41, G42) and the offset signs.

Offset value sign / G code	+	−
G41	Left offset	Right offset
G42	Right offset	Left offset

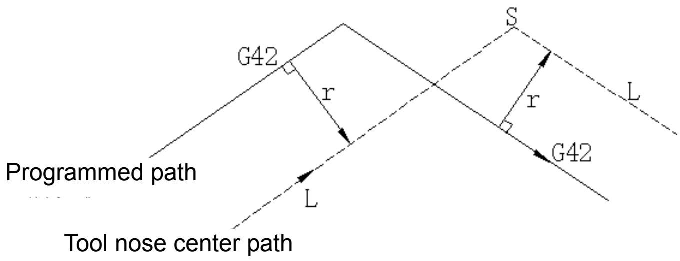
In some exceptional occasion and in offset mode, the offset direction can be changed. But, the block offset direction of a starting block and a block following the starting block can not to be changed.

When changing the offset direction, the inner side and outer side concepts cannot be considered.

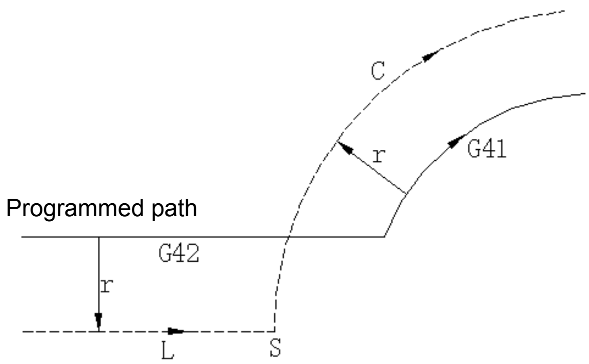
The following explanations show the case when G41 or G42 are employed to change the offset direction. When it is changed by the offset value sign, see Section 2.6.3.2.3.

Supposing the offset value sign is positive in the following illustration.

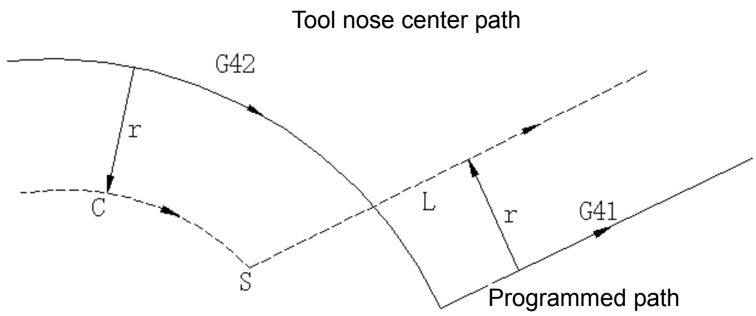
(I) Linear to linear



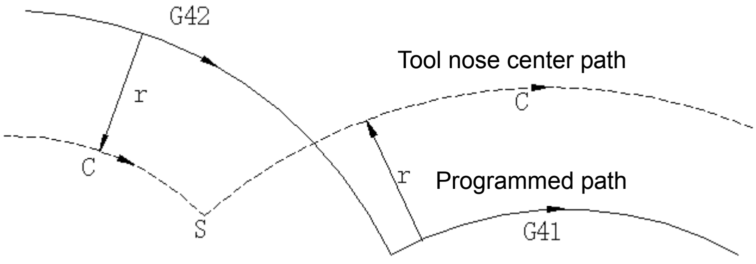
(II) Linear to arc



(III) Arc to linear



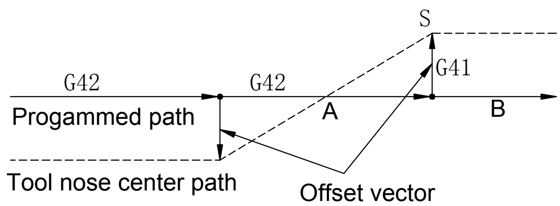
(IV) Arc to arc

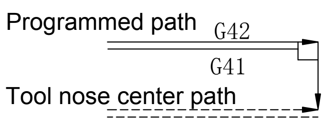


(V) Tool movement when an offset is normally performed without intersection

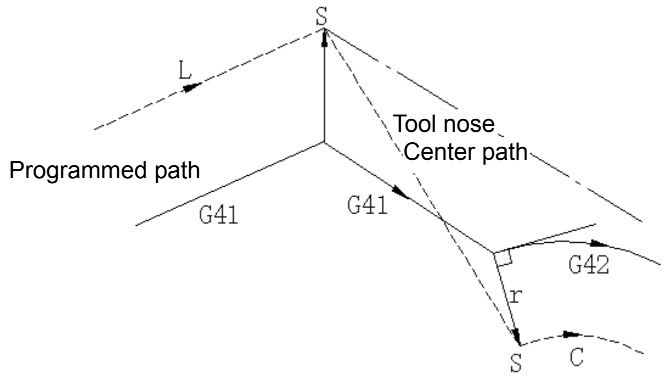
When offset direction is changed with G41 and G42 from block A to B, if the intersection with offset path is not required, then the corresponding vector will be generated at the starting point of block B.

Linear→linear

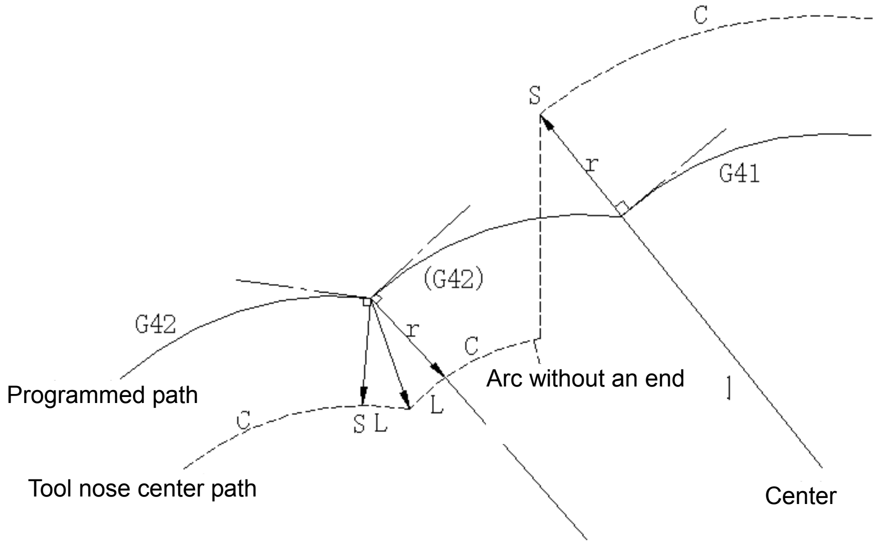




Linear to arc



Arc to arc



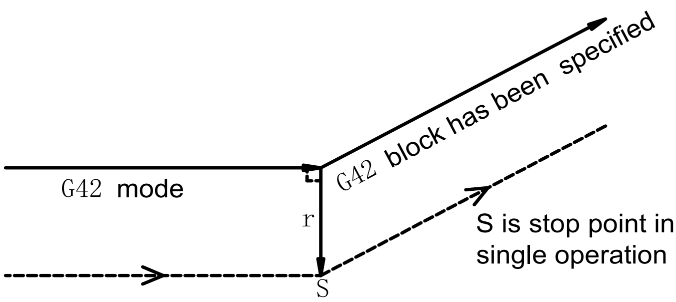
（6）Tool nose radius compensation G code in offset mode

A vector of upright to the previous block is formed relative to movement direction when the tool nose radius compensation G (G41, G42) is specified in offset mode, which is not related to inner, outer side machining.

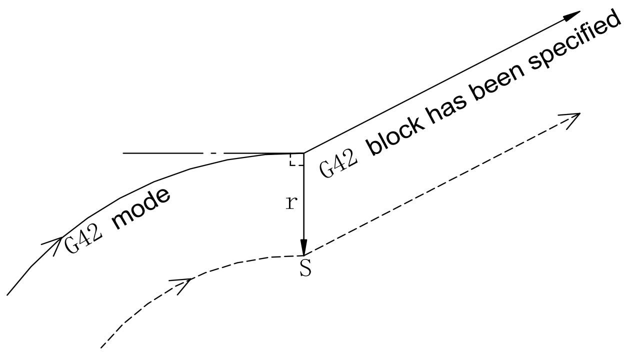
If this code is included in the arc command, and then the arc movement can not be correctly applied.

When the offset direction is changed by the tool nose radius compensation G code (G41, G42), Section 2.6.3.5 is referred.

Linear to linear



Arc to linear



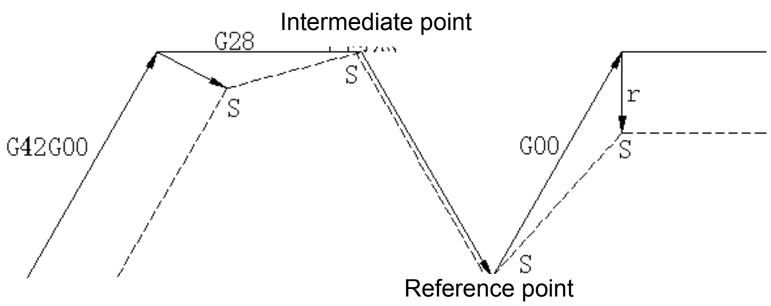
（7）Temporarily cancelling offset

If the following commands are specified in offset mode, the offset is temporarily erased, and then the system is automatically restored to offset mode.

See Section 2.6.4.2 for offset cancel and star.

(a) G28 returns to reference position automatically.

If the G28 is specified in offset mode and the offset in the intermediate point will be erased, the offset mode will then automatically restored after the reference position return is done.

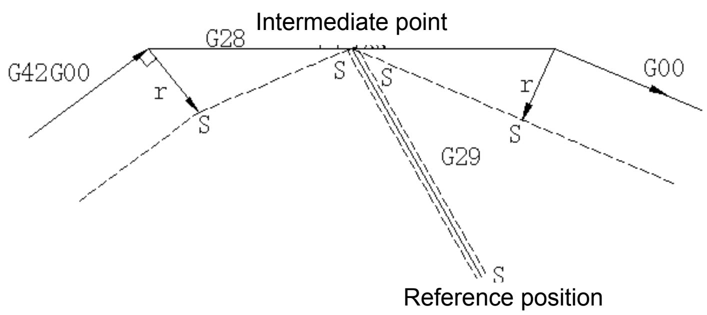


S point stops in single block operation.

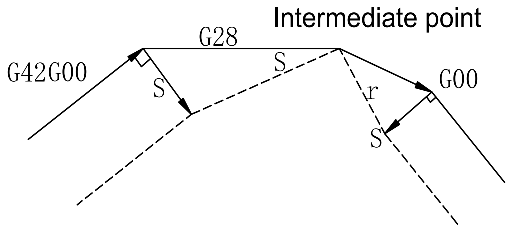
(b) G29 automatically returns from reference position.

If the G28 is specified in offset mode and the offset in the intermediate point will be cancelled, the next block will restore the offset automatically.

(I) When G29 is immediately specified after G28



(II) When G29 is not to be followed immediately after G28

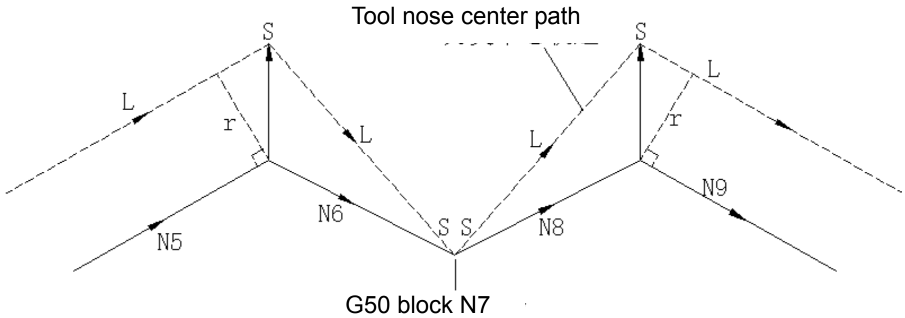


（8）Commands for cancelling vector temporarily

When the following commands are specified in offset mode, the offset vector is temporarily cancelled. Then the offset mode is automatically restored.

In this case, there is no offset cancel movement. Tool nose center arrives programmed point from the top of vector of offset paths intersection point. When system enters offset mode, tool path center arrives directly to the top of vector of offset paths intersection point.

（a）Workpiece coordinate system setting（G50）



（G41 mode）

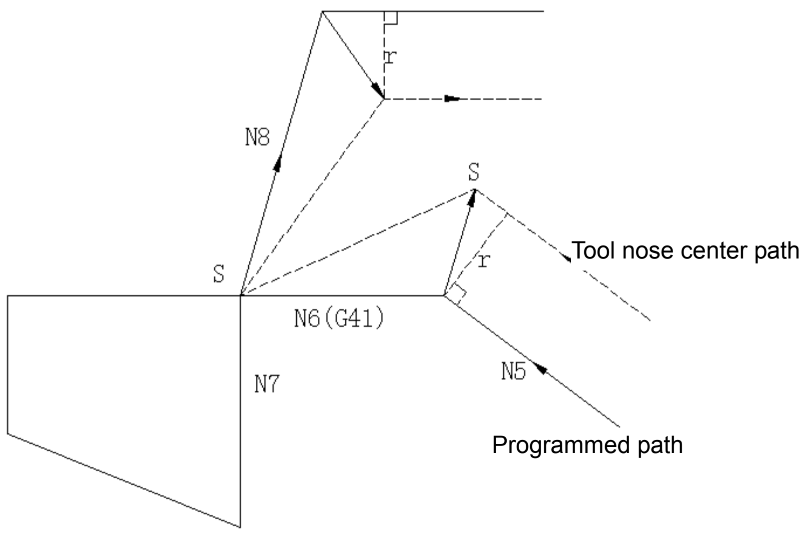
N5 G01 U3000 W7000;

N6 U-3000 W6000;

N7 G50 X1000 Z2000;

N8 G01 X4000 Z8000;

（b）canned cycle（G90,G92,G94）and multiple cycle（G71～G76）



（G42）

N5 G01 U5000 W-6000；

N6 W-8000；

N7 G90 U-6000 W-8000 I-3000；

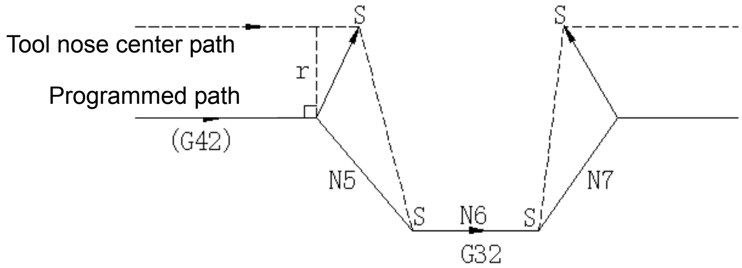
N8 G01 U12000 W5000；

For details of too nose radius compensation in canned cycle, see the following Sections:

G90, G94　　See Section 6.2.5.2

G70～G76　　See Section 6.2.5.3

（c）G32 thread cutting



Too nose radius compensation includes G32 block is disabled.

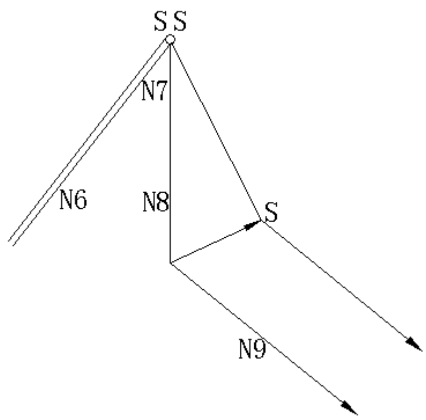
（9）Block without tool movement block

The following blocks do not specify the tool movement and the tool will not move even if the tool nose radius compensation is executed in the following blocks.

① M05;	M code output
② S21;	S code output
③ G04 X1000;	Dwell
④ G01 U0;	Movement distance 0
⑤ G98;	G code only
⑥ G22 X1000000;	Workpiece area setting
⑦ G10 P01 X10 Z20 R10 Q1;	Offset value changing

（a）It is specified in starting state

The block which does not specify the tool movement is input when starting, an offset vector does not generate.



G04………

N6 U1000.0 W1000.0;

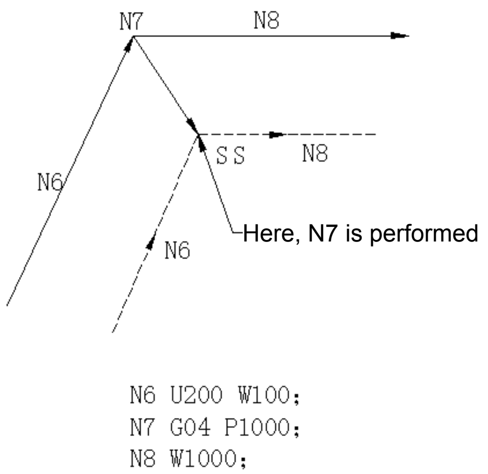
N7 G41 U0;

N8 U-1000.0;

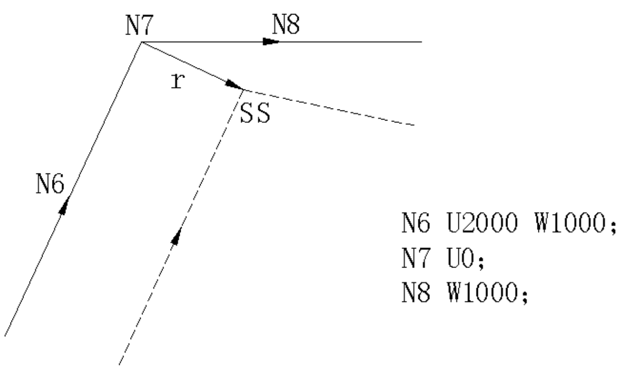
N9 U-1000.0 W1000.0;

（b）Specify blocks without tool movement in offset mode

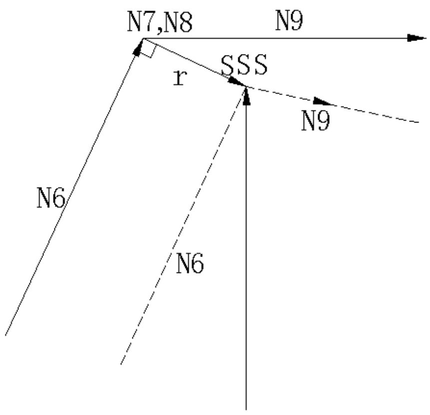
When a block without tool movement is input in offset mode, offset vector and tool nose path are the same when this block is not specified. (See the offset mode in Section 2.6.4.2)



But, when the block movement distance is 0, even if the block is separately specified, the tool movement becomes the same of that of two blocks or more without tool movement command. (This will be explained in the following)



More than two blocks without tool movement cannot be commanded serially. If done, the position upright to the previous block starting point is taken as the end point of current block, and then the overcutting will be generated in the vertical direction of tool movement.



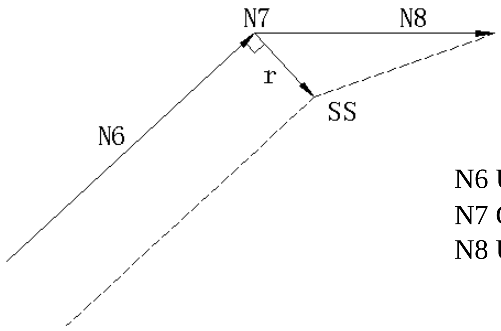
N6 U200 W100;
N7 S21;
N8 G04 U100;
N9 W1000;

Here, N7, N8 are performed

SSS, is depicted that the tool stops for 3 times, when single block operates.

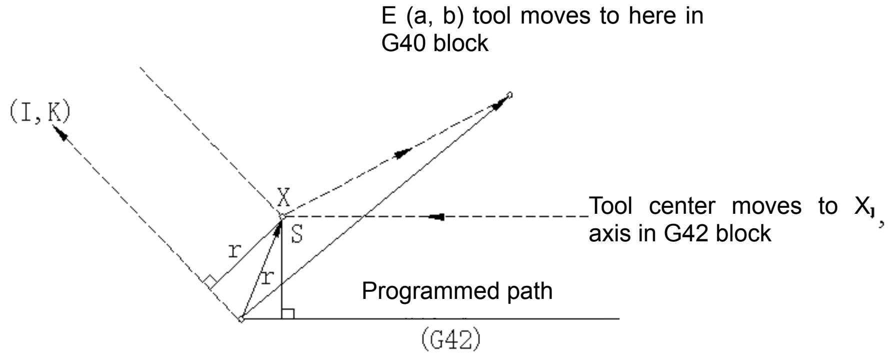
(b) When offset cancel command is specified.

When the block without tool movement is gather together with offset cancel command. A vector in the vertical direction of the previous block tool movement is generated, its length is equal to the offset value, and this vector is cancelled in next command.



N6 U1000 W1000;
N7 G40;
N8 U0 W1000

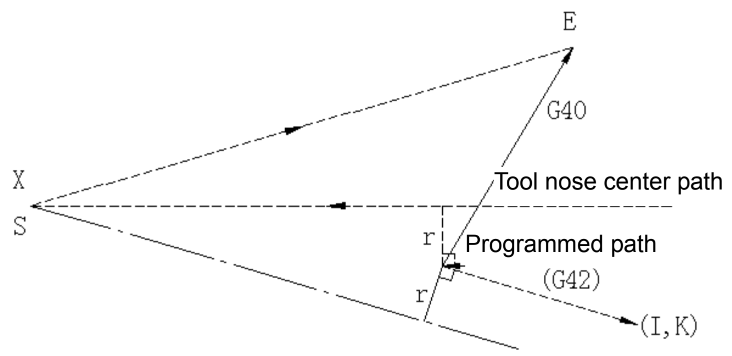
（10）Blocks with G40 and I,K



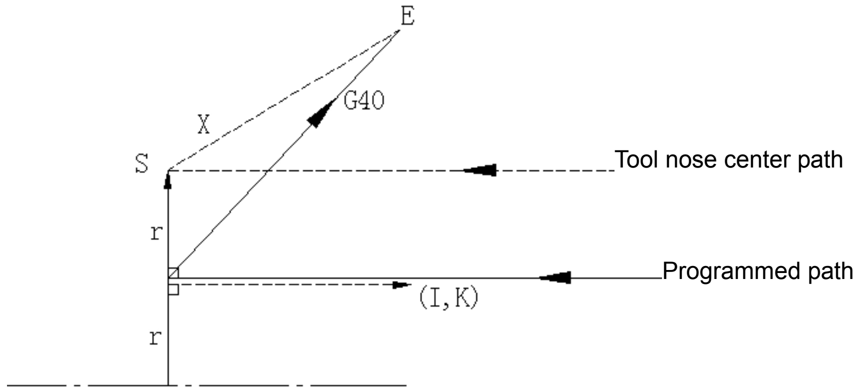
（G42 mode）

G42X$\frac{a}{-}Z\frac{b}{-}I$ -K;

In this case, note that the NC gains an intersection regardless of the inner side or outer side machining.



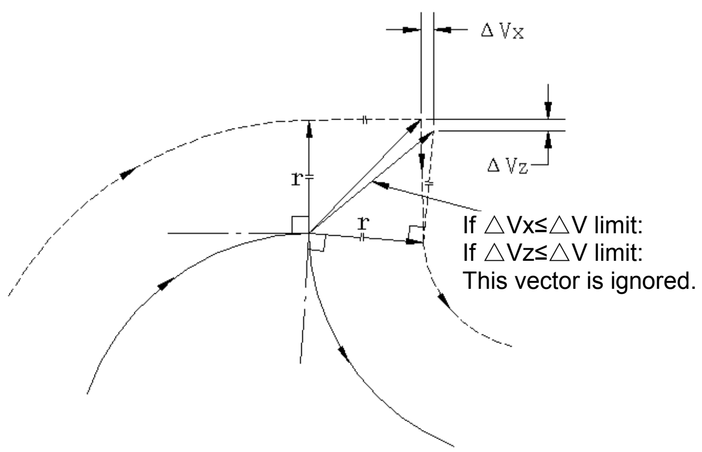
When an intersection can not be gained, the tool moves to the normal of previous block at the end point of the previous block.



（11）Corner movement

When two or more vectors in the end of a block are generated, tool vector moves to another with linear.

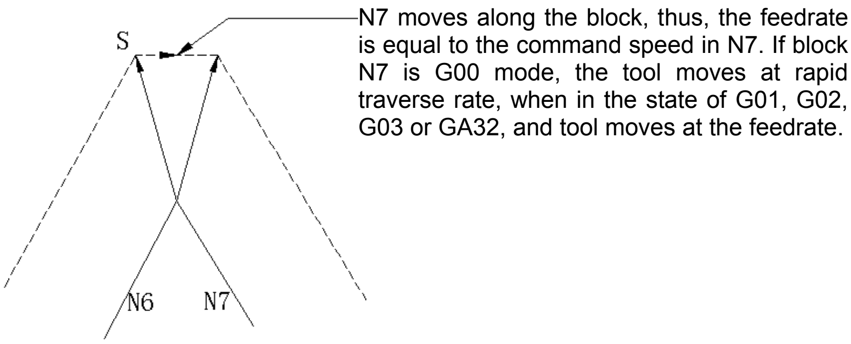
If almost these vectors are consistent, and the chamfering is then not operated and the following vector will be ignored.



If △Vx≤△V is limit, △Vz≤△V is limit, the following vector is ignored, △V limit is preset in the NC unit. The △V limit value is specified with the parameter (CRCDL, No.69).

When these vector do not coincide, a movement around corner occurs, which movement

belongs to the following block.



（12）Interference check

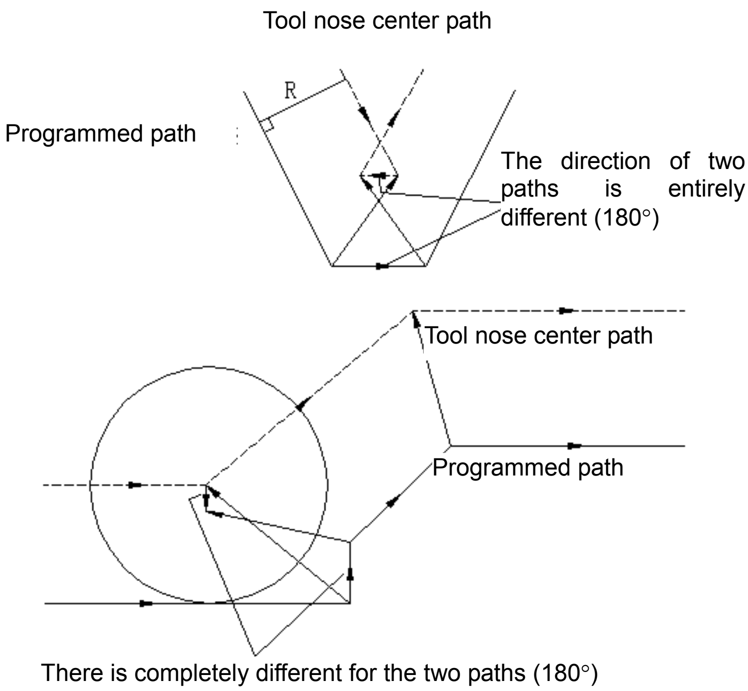
Tool overcutting is called as “Interference”. Interference check function checks tool overcutting in advance. However, not all the interferences can be checked by this function. Interference check should be performed even if the overcutting never occurs.

（a）Reference condition of interference

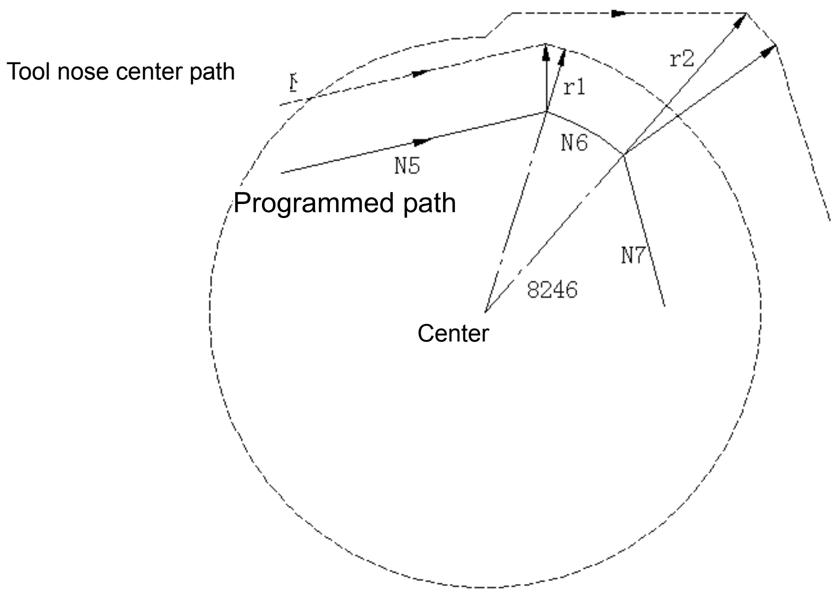
① The direction of tool nose center path is different from programmed path in too nose radius compensation. (The path between 90° and 270°)

② Except above conditions, in arc machining, the angle of start and end points of tool nose center is entirely different from that of start and end points of programmed path (it is more than 180°) .

Example for condition ①



Example for condition ②



N5 G01 U2000 W8000 T11;

N6 G02 U-1600 W3200 I-8000 K-2000 T22;

N7 G01 U-5000 W2000;

（r_1＝2000，it is cutter compensation value of T11）

（r_2＝6000，it is cutter compensation value of T22）

In above illustration, the arc in block N6 is in the first quadrant. The arc passes 4 quadrants after the cutter compensation is applied.

(b) Interference modification in advance

① Delete the vector for causing the interference

When the tool nose radius compensation is performed, vector V_1, V_2, V_3 and V4 are generated between A and B, V_5, V_6, V_7 and V_8 are generated between B and C. Checking the near vector at first, if the interference is issued, they will be ignored. But, if the vector will be ignored because of interference is the corner last vector, they can not be ignored.

Interference check between V_4 and V_5 → Interference →V_4 and V_5 are ignored.

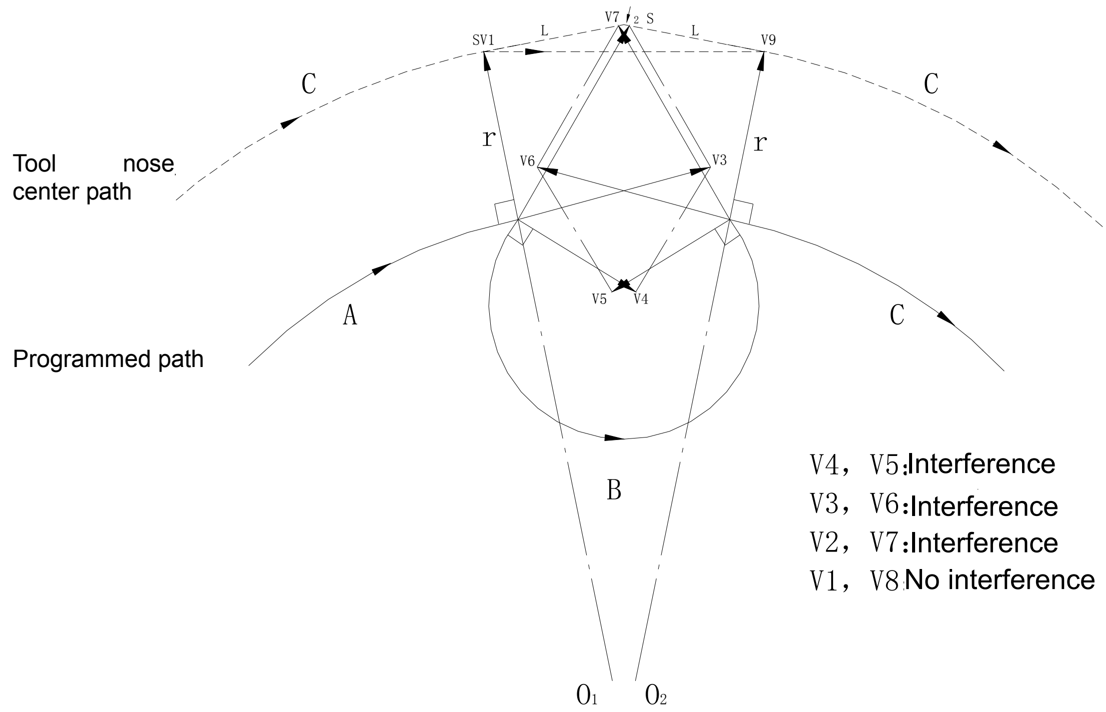
Interference check between V_3 and V_6 → Interference →V_3 and V_6 are ignored.

Interference check between V_2 and V_7 → Interference →V_2 and V_7 are ignored.

Interference check between V_1 and V_8→ Interference →V_1 and V_8 should not be ignored.

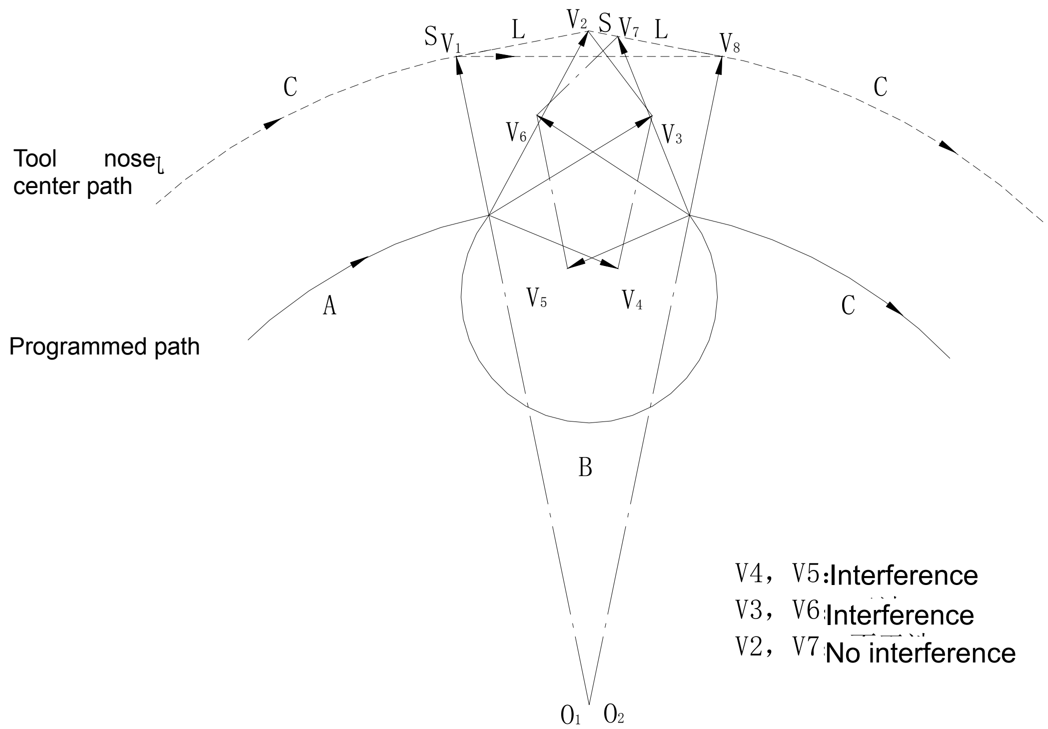
A vector without interference is checked when the check executes, the following vector will not be checked. If block B is arc movement, this vector is interference vector will generate a linear movement.

(Example 1) Tool linear movement from V_1 and V_8



Note: When block A is executed in Single block mode and the tool stops, the tool center moves to V3.

(Example 2) Tool linear movement is as follows:

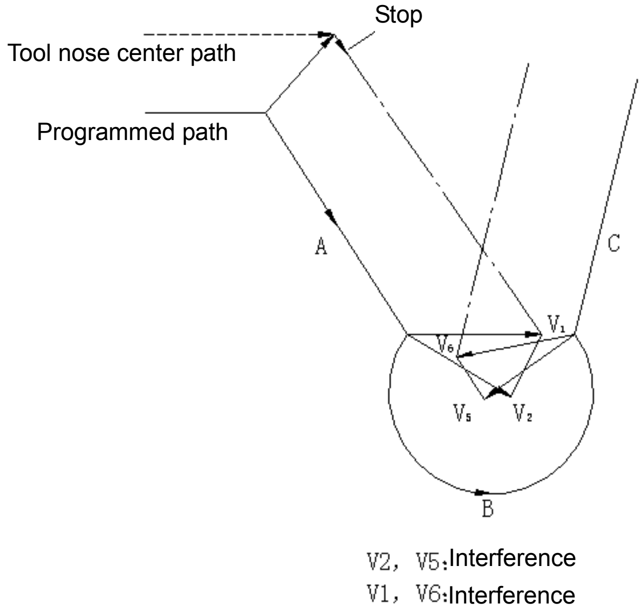


② If interference is generated after dealing with ①, tool stops with an alarm.

If an interference is generated after dealing ① or there is only one pair vectors from the beginning of check and this pair vectors are interfered. An alarm (N041) displays and tool

immediately stops after the previous block is performed.

(If single block operation performs, the tool then stops at the end of block).

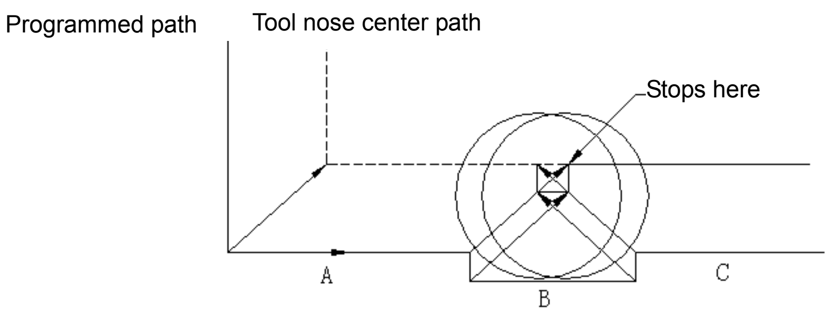


Vector V_2 and V_5 are ignored due to the interference, and then the interference is generated between V_1 and V_6. The alarm displays and tool stops.

(c) when checking the interference, it is actually not generated.

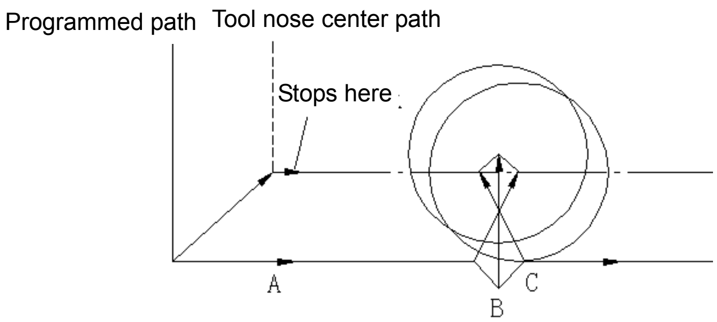
Example is as follows:

①Depth is less than tool nose radius.



Although the interference is not generated actually, yet due to the direction of tool path is different from programmed path, the No. 41 alarm generates and tool stops.

② Depth of groove is less then tool nose radius.

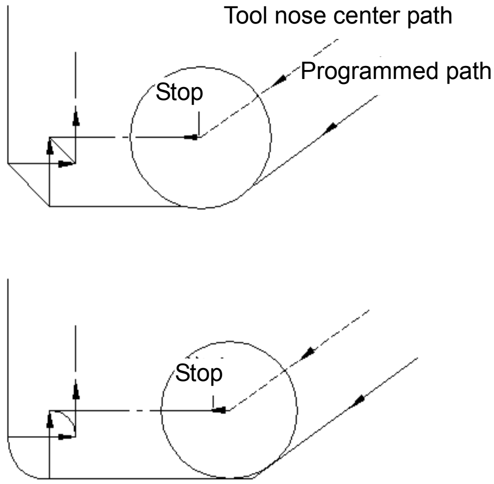


The direction of tool path is different from programmed path. (Same as the illustration ①).

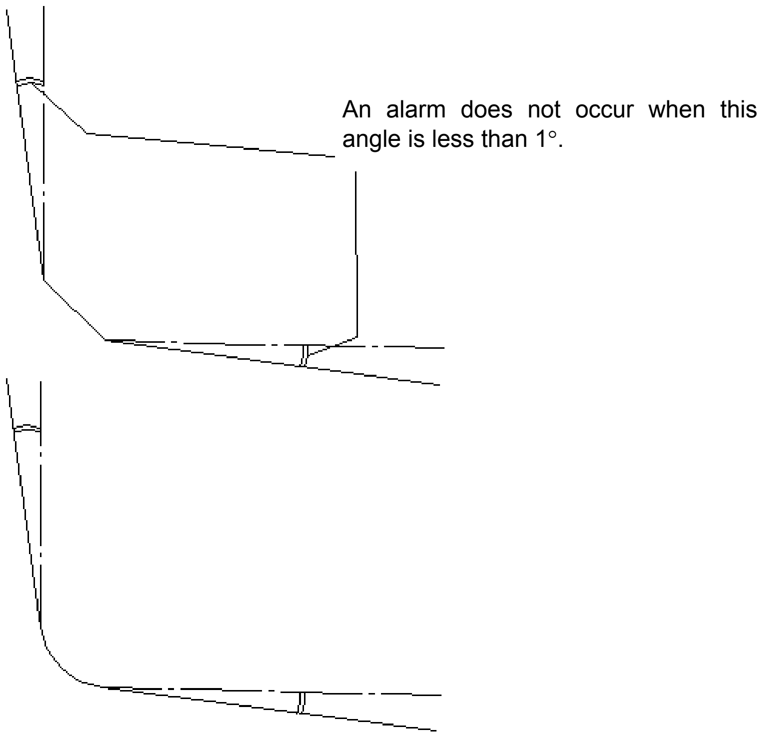
(13) Deal with the chamfering and corner.

(a) When chamfering or corner arc applies and when the corner exists and interference ignores, the cutter compensation executes.

(b) In offset cancel mode, a start block or when the offset direction changed, the compensation is not performed, and the tool stops and an alarm occurs (No039).

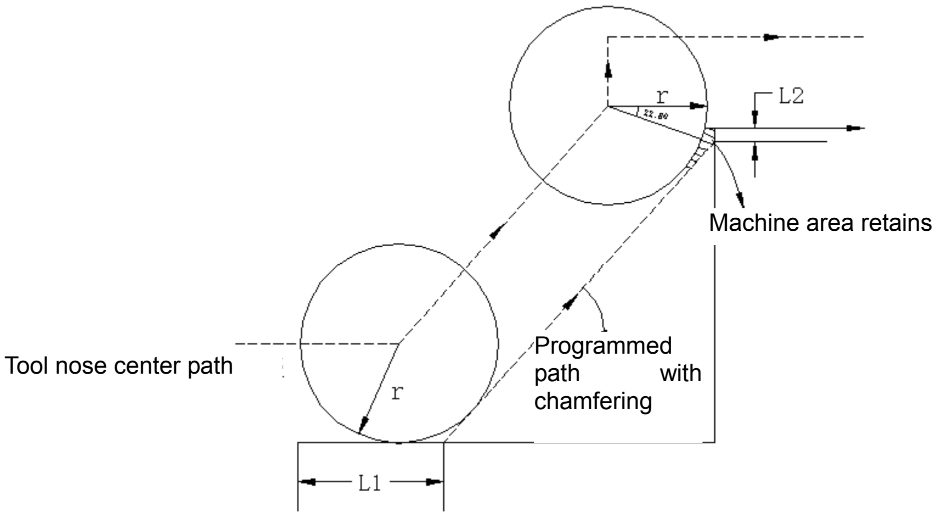


(c) If the programming obliquity is less than 1°, P/S (No.52 and No.54) alarms will not be caused by the error of cutter compensation.



(d) When the machine area residues or an alarm occurs

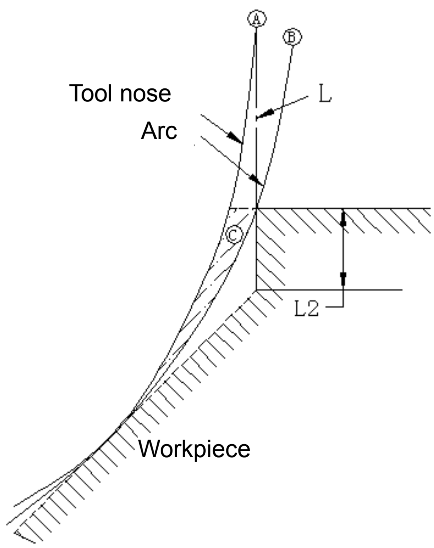
(Ⅰ) The following illustration indicates that the cutting in machine area is not adequate.



The section（l_1 or l_2 in the above figure) of programmed path other than chamfering is in the following range in inner corner, the short cutting may occur.

0≤l_1 or l_1≤r tan22.5°（r：Tool nose radius）

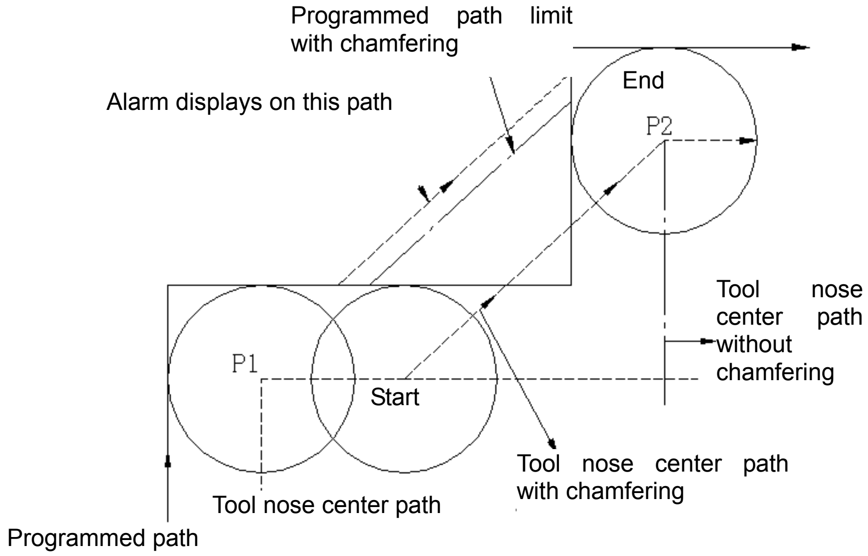
The enlarged drawing of the machining remains part



Although the tool can be positioned at B of the above figure, yet it is still positioned at A (Tool nose is tangent to line L).

In this case, area C is not machined.

(II) No.52 and No.55 alarms occur in the following circumstance.



The outer chamfering with offset is increased a limit to programmed path.

In the above illustration, the start/end points of tool nose center path is separately consistent with the intersection（P1 or P2） which is without chamfering, the outer chamfering is limited.

When chamfering path is consistent with the intersection (P1 or P2) without chamfering, the outer chamfering is limited. When the chamfering value is more than the specified limit value, No52 and 55 alarms occur.

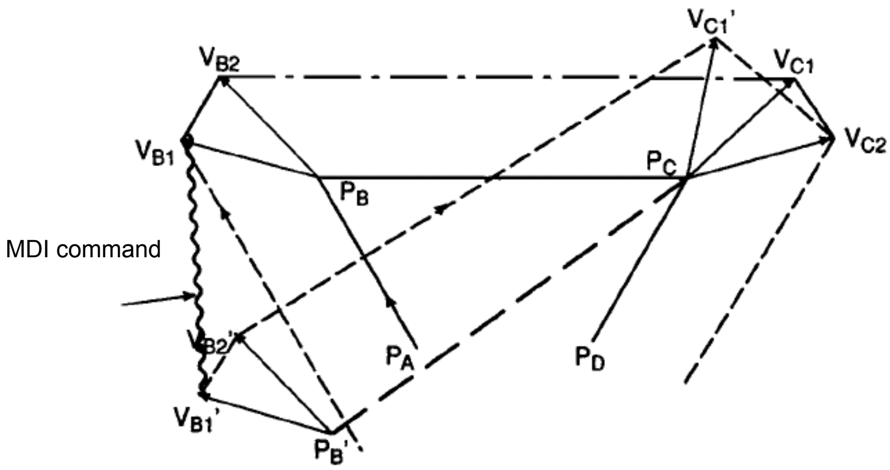
（14）MDI inputs command

The command which is input from MDI does not perform the tool nose radius compensation.

In automatic operation mode, when the NC program specified by absolute value is performed, and the single block function temporarily stops, the MID operation executes, the automatic operation is then restarted, and the tool path is as follows:

At the moment, the vector at the start of next block is shifted and the other vectors will be generated by the following two blocks. So the offset after point P_c can execute the tool nose radius compensation.

In this case, the vector at starting point of the next block executes translation and the following two blocks generate other vectors. The following 2nd block (point P_c in the following figure) executes the tool nose radius compensation.



Tool nose center path

When point P_A, P_B and P_C are programmed with absolute command, after the block is performed from P_A to P_B and the tool stops by single block function to move tool by MDI operation, the vectors V_{B1} and V_{B2} are transferred to V_{B1}' and V_{B2}', and the vectors V_{C1} and V_{C2} between block P_B-P_C and P_C-P_D are recalculated.

However, because the vector offset V_{B2}' is not recalculated anymore, the accuracy compensation can be performed from point P_C.

(15) Manual operation

Refer to Section 4.3.4 when manual operation is executed in tool nose radius compensation mode.

(16) Subprogram

The system must in offset cancel mode before calling subprogram (before executing M98). Offset is enabled after subprogram is entered, but the system must be in offset cancel mode before returning a main program, otherwise, No. 036 alarm occurs.

(17) Notes

(a) Specify offset value

The specified number following address T is as follows:

T (□ + □)

Tool offset number

Tool number

As for T(2+1) or T (2+2), both two modes can be set, the tool offset number lower one digit or two digits of T code are specified the offset value, in this case, the offset value corresponding to the tool offset number is employed (Tool nose radius).

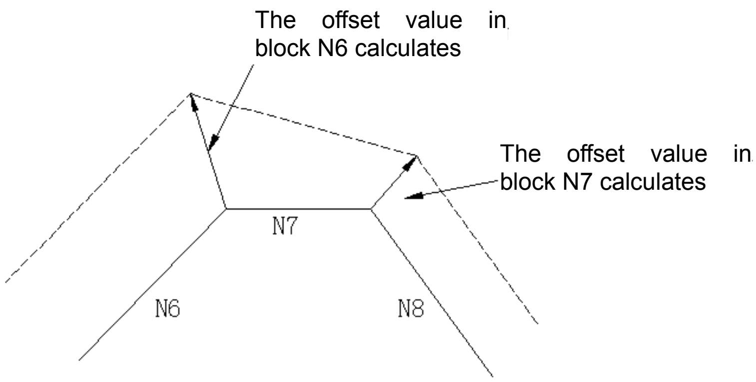
Once the offset number is specified, it keeps valid until the other offset number is specified or the offset mode is cancelled.

Besides the offset number specifies the offset value for the tool nose radius compensation, it is also used to specify the offset value for tool offset use.

See Section 2.6.1.

(b) Offset value change

Normally, the offset value changes in cancel mode or tool change. If the offset value in offset mode is changed, the new offset value will calculate new vector in the end of block, and the image tool number and tool offset number can also be changed with the same mode.



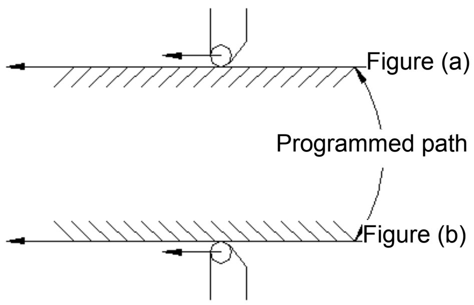
When some vectors between block N6 and N7 are generated, the offset value of block N6 calculates the vector in the end of current block.

（c）Offset amount polarity and tool nose center path

When the negative offset value is specified, the program in the figure is performed by changing G41 into G42, or change G42 into G41.

When tool machine is applied in inner side and it will become into outer contour machining, vice versa.

Example is as follows: usually, supposing the NC machining program is positive offset value. When a tool path (see the figure a) is specified in program, if a negative offset value is specified, tool must be moved according to the (b); the tool movement on (b) will become into (a) when the offset value is reversed.



Note: Tool nose offset vector reversed depending on the sign of offset value, but the direction of tool nose supposed is invariable. Therefore, when the supposed tool nose is machined at the starting point, the sign of offset value can not be changed!

2.6.5 Offset Value Input with G10

The offset value can be entered from program when using the following commands.

G10 P-X-Z-R-Q-; or

G10 P-U-W-R-Q-;

P: Offset number

X: Offset value along X axis (Absolute)

Z: Offset value along Z axis (Absolute)

U: Offset value along X axis (Incremental)

W: Offset value along Z axis (Incremental)

R: Tool radius offset value (Absolute)

Q: Imaginary tool nose number

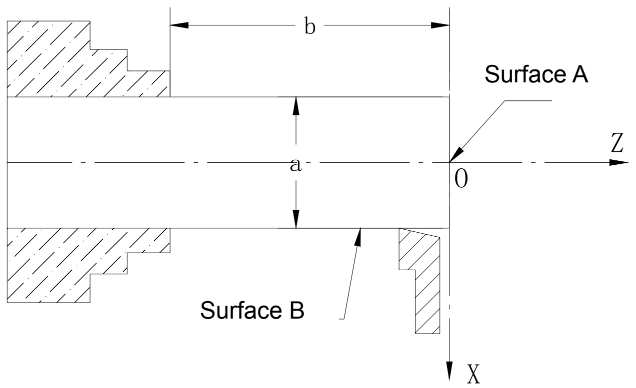
The value specified in X and Z axes are regarded as offset value setting by with absolute command, it corresponding to the offset number specified by address P.

The value is specified with address U and W which is added to the current offset value for the corresponding offset number when it is specified with incremental value.

Note 1: Address X, Z, U and W can be specified at the same block.

Note 2: By using this command in block, the tool makes fewer advances; this command is also used to input an offset value once, the command is continuously specified and enter these values instead of MDI. (One by one)

2.6.6 Directly Input Tool Offset Value by Manual Measure (Trial-Cut Tool- Setting)



Supposing the workpiece coordinate system is set in advance. (See Section 2.3.5 and Section 2.5.8)

(1) Start the spindle with manual mode, operate once with tool along the workpiece surface A. Press position record key, the tool retracts and the spindle stops.

(2) Measure the workpiece length b, select the tool figure offset page, type "N101" (or, the current tool number corresponding to the offset number N102 or N107 are also input). Then input key is pressed, "Z" and the measured workpiece length b can be input after the "N" flashes at the lower left corner, (If the Z axis coordinate zero point is regarded as workpiece surface, so the Z0 can be input) and the input key is controlled again.

(3) Start the spindle, operate once along the workpiece surface B. The "Position record" key is controlled, the tool retracts and spindle stops.

(4) Measure the workpiece diameter "a", select the tool figure offset page, typing N101 (or, the current tool corresponding to the offset number N102 or N107 can be input). The input key is pressed, "X" and the measured workpiece diameter value "a" can be input after viewing the letter "N" flashes, and then, the input key is pressed again.

(5) All the measured data is input with decimal point

Note: Tool figure offset page.

Tool figure offset 01: 00001 N0001

NO	X	Z	R	T
G01	000.000	000.000	000.000	0
G02	000.000	000.000	000.000	0
G03	000.000	000.000	000.000	0
G04	000.000	000.000	000.000	0
G05	000.000	000.000	000.000	0
G06	000.000	000.000	000.000	0
G07	000.000	000.000	000.000	0

Current position: (Relative coordinate)

U 121.030 W 214.450

(Worn offset) (Workpiece offset) (Figure offset) ( ) ( )

2.6.7 Automatic Tool Offset Input

The tool moves to the measure position when a command is executed, CNC automatically checks the difference between the current coordinate value and specified coordinate value, and it is regarded as the tool offset value. When the tool moves to the measure position in offset mode, as for the resultant, the further offset value generates, the current offset value is further offset.

（1）Coordinate system

A coordinate system G50Xx Zz should be set beforehand before the tool moves to the measure position; this coordinate system is then set (Using common workpiece coordinate system when programming).

(2) Moving to measure position

G36 Xxa Zza is specified to move to the measure position in MDI or Auto mode. In this case, the measure position should Xx or Zz (Absolute command).

The tool moves to the measure position at rapid traverse when the this command is executed, the feedrate is reduced when approaching measure position, and then the tool moves continuously till to the end signal (from measure equipment) issues. When tool nose approaches the measure position, the measure equipment sends a signal to NC to stop the tool.

(3) Offset

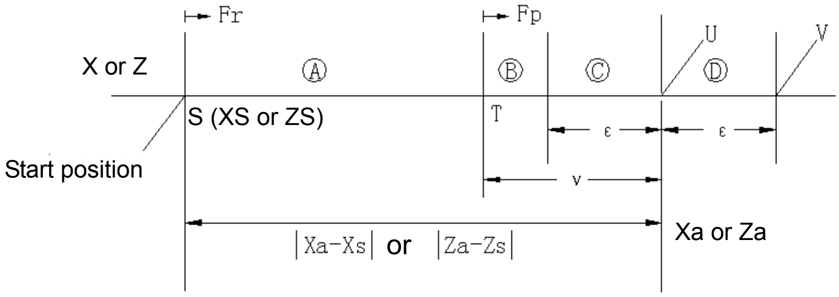
When tool arrives measure position (α, β) when the value of Xa or Za is specified using G36 Xxa or G37 Zza, the current tool offset value is further replaced by the D-value between the coordinate value (α or β) and Xa or Za.

Offset value X = current offset value X + (α－Xa)

Offset value Z = current offset value Z +(α－Za)

These offset values can also be changed from MDI key-board.

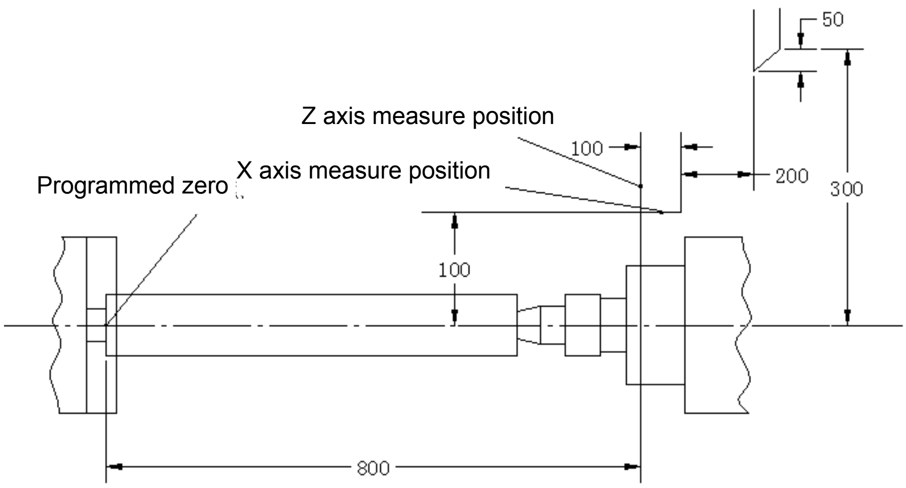
(4) Feedrate and alarm



Fr: Rapid feedrate
Fp: Measure feedrate

When the start position moves to G36 or G37 which the measure position is specified by Xa or Za in advance, tool overruns the area and feed at the rapid traverse. And then, the tool stops at point (Xa－vx) or (Za－vz) and over the B, C and D area and move at feederate set by parameter in measure. If the arrival end signal is switched on when crossing area B, the No.080 alarm occurs. If the arrival end signal is not switched on before point V, tool stops at point V and the No.080 alarm also stops.

Program example



	Offset amount (Before measuring)	Offset amount (After measuring)
X	100000	98000
Z	0	4000

G50X960000Z110000；Absolute zero program (Coordinate system setting)

S01 M03 T0101; Specifying tool No.1, offset No.1 and spindle revolution

Z850000; Move to measure position

(If tool arrives to the check position in X19800; because the 200mm is correct measure position, so, the offset value should be modified into 198.0-200.0=-2.0mm).

G00X204000; Fewer retraction along X axis

G37Z800000; Move to Z axis measure position

(If tool arrives to the measure position in Z804000; the offset value changes into 804.0-800=4mm)

T0101; It is displaced by the D-difference; the new displacement is valid when specifying the T code anew.

Note 1: No.081 alarm occurs when T code is not specified before G36 or G37.
Note 2: No.82 alarm occurs when T code is specified and shares the same block with G36 or G37.

（5）Offset value

There are 16-group offset numbers (01~16), also, you can extend to 32-group.

Note 3: Measure speed v and ε are regarded as parameter setting (refer to (4) for v, ε value). These values are set by machine tool builder.

Note 4: v and ε must be a positive, v＞ε.
Note 5: Watching the measure position arrival signal each time 2ms. The following measure error generates

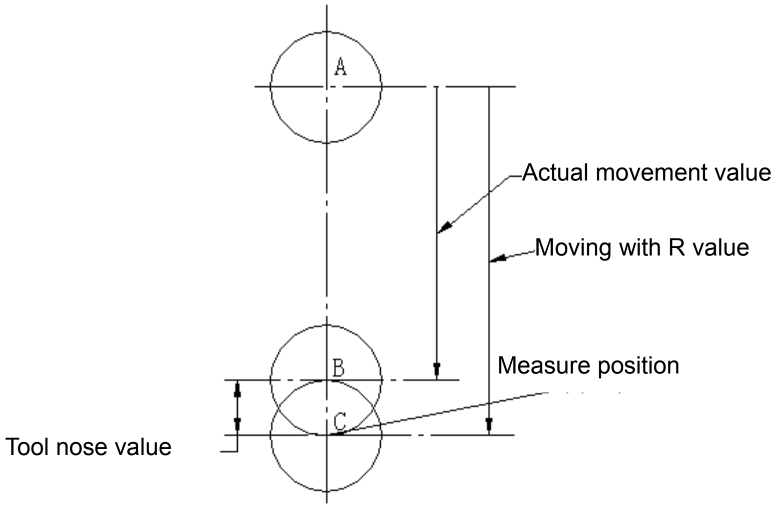
ERRmax=Fm×1/60×2/1000

Where: ERRmax: the max. measure error (mm)
Fm: Measure feedrate (mm/min)
Note 6: Erasing cutter compensation before specifying G36 and G37.
Note 7: When manual movement is inserted to the movement of measure feedrate, tool returns to the previous position which the manual insertion restarts.

Note 8: Tool stops after lasting 16ms at most and after the measure closes to signal, when a coordinate is closed to the signal by measuring (This coordinate is not for machine stop), offset value are determined by α and β (see (3), the corresponding 16ms stroke is: Qmax＝Fm×1/60×16/1000.

Where: qmax: the max. stroke value (mm)
Fm: Measure feedrate (mm/min)
Note 9: When an optional cutter compensation function is employed, the tool offset value is determined by tool nose R, it is sure that the tool nose radius can be correctly set.

For example: When the tool nose center is consistent with start point.



The tool moves actually from point A to B to decide the offset value, and the R value should be considered when tool moves to the point C.

Note 10: When this function is employed, setting the parameter 008-TLCC to 0. TLCC 0: the next T code is valid when the offset amount changes.

2.6.8 Outer Tool Compensation

This function can be changed the compensation value from the outer device, when entering the compensation value from outer device, the data entered is increased to the offset value of specified offset number in a block. If it is specified with entering signal, the input data will be formed of offset value.

If the machine with automatic measure tool and workpiece device, the offset value offers the modification value to NC with offset value. The offset value is offered the NC modification offset value.

The program, operation, function and restriction procedure are different for different machine tool builders. Refer to the manual issued by the machine tool builder when operating.

Note: When the offset number is specified to 0 in a block, if the cutter compensation inputs from outer equipment, that is to say, when the offset is erased, the workpiece coordinate system can be moved to input value (it is determined by the machine), refer to the appropriate manual issued by the machine tool builder.

2.7 Cycle Machining Function

The repetitive machining is the especial function for the turning machine, as rough-turning. Normally, a piece of serial path needs to describe by three or more than ten commands, a command can be specified to use for cycle function. Besides, the variable value related with repetitive motion needs to prescribe again, simplifying the program by using these canned cycle and it will more valid

Generally, the canned cycle includes two kinds

(1) Single canned cycle

(2) Complex canned cycle

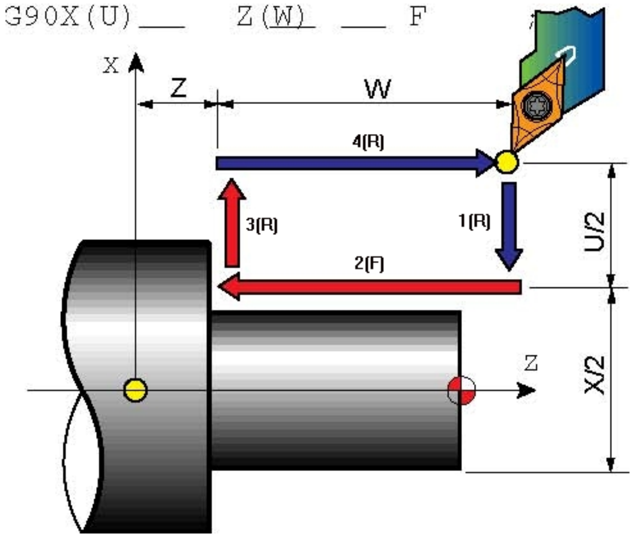
Note: The diameter programming is used in the following figures. When programming, the tool start point is set outside the machining path, not inside.

2.7.1 Single Canned cycle (G90, G92 and G94)

Three canned cycles G90, G92 and G94 are available.

2.7.1.1 Outer Cylindrical Surface Cutting Cycle(Axial Cutting Cycle)—— G90

Linear cutting canned cycle is employed in the following commands.

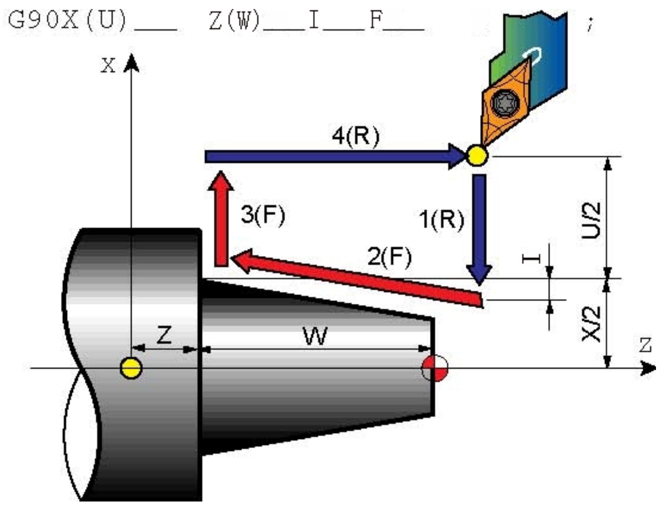


R…rapid traverse

F…federate specified by F code

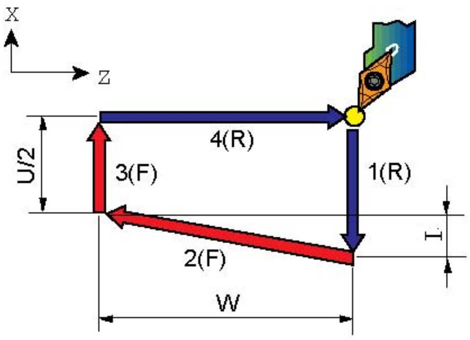
The positive/negative sign of digital followed address U and W is determined by direction of path 1 and 2 in the occasion of incremental programming. They are negative in the above illustration, the 4 steps in the above figure are finished by pressing CYCLE START key again and again in a single block mode.

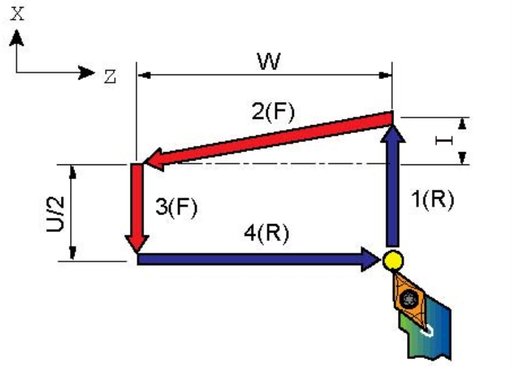
(2) Taper machining cycle can be described by using the following command



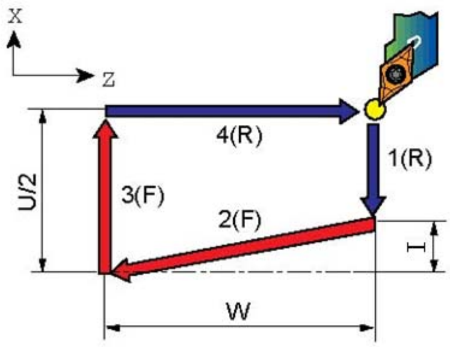
With respect to the incremental programming, the positive/negative sign of digital described by the address U, W and I have the following relations with tool path:

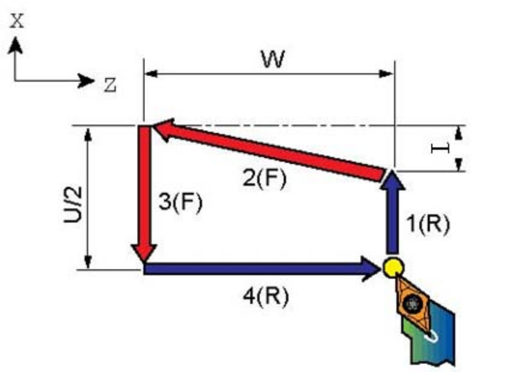
（ i ）U<0，W<0，I<0　　（ii）U>0，W<0，I>0



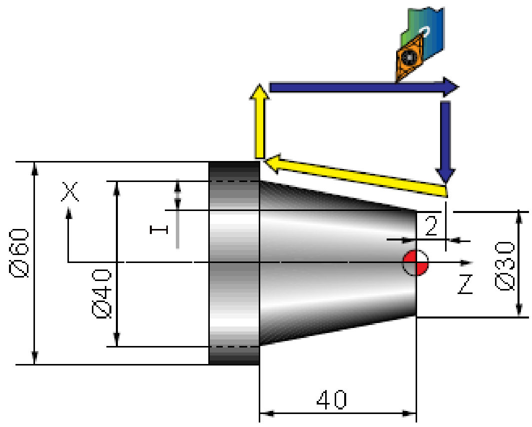


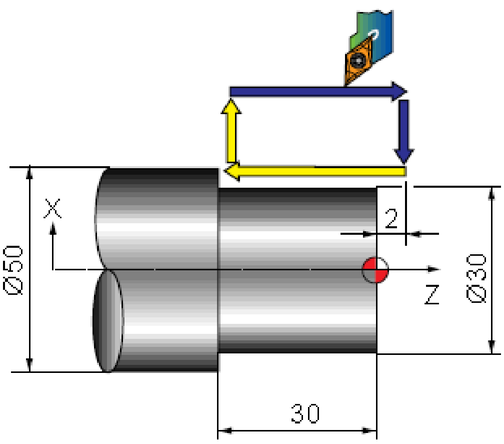
（iii）U<0，W<0，I>0，$|I| \le \left|\frac{U}{2}\right|$　　　　(iv)U>0，W<0，I<0 ，$|I| \le \left|\frac{U}{2}\right|$





Operation:






```
G30 U0 W0 ;
G50 S2000 T0100 ;
G96 S200 M03 ;
G00 X61.0 Z2.0 T0101 M8 ;
G90 X55.0 W-42.0 F0.25 ;
X50.0 ;
X45.0 ;
X40.0 ;
Z-12.0 I-1.75 ;
Z-26.0 I-3.5 ;
Z-40 I-5.25 ;
G30 U0 W0 ;
M30 ;
```

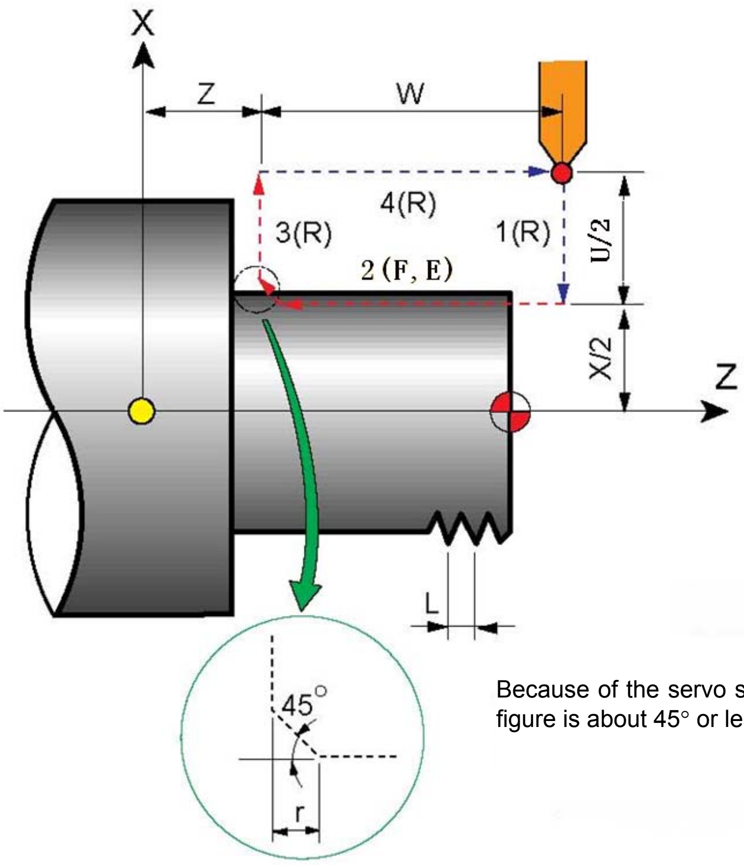
```
G30 U0 W0 ;
G50 S2000 T0100 ;
G96 S200 M03 ;
G00 X56.0 Z2.0 T0101 M08 ;
G90 X51.0 W-32.0 F0.25 ;
X46.0 ;
X41.0 ;
X36.0 ;
X31.0 ;
X30.0 ;
G30 U0 W0 ;
M30 ;
```

2.7.1.2　G92 Thread Cutting Cycle—— G92

The tool rapidly traverses from starting point to thread' starting point, cuts the thread surface, rapidly retracts to thread surface, at last rapidly returns to the starting point, which cut path is a closed path. For multi-thread, such cycle cutting is executed. When there is a consecutive command following G92 for deep cut again, cycle cutting again is executed. Using consecutive command gradually increases cutting deep amount, and deep thread cutting can be executed.

（1）Linear thread machining describes by using the following command

G92X（U）——Z（W）—— { F—— / E—— } ；



R…Rapid traverse

F…Description with F code

E…Description with E code (Selection)

L: Thread lead

Because of the servo system delays, the angle in the left figure is about 45° or less than 45°.

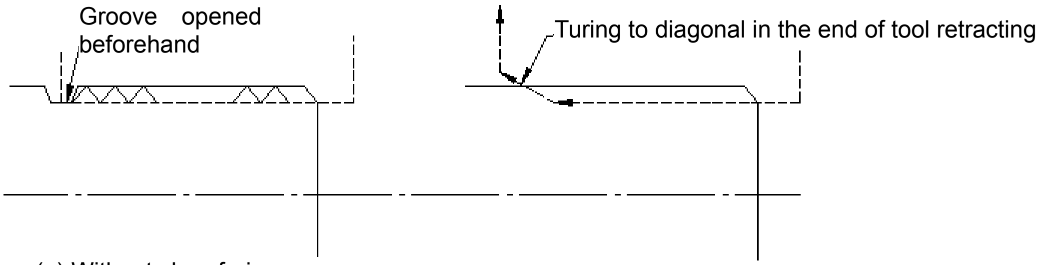
The sign of digital followed address U and W is relative to the direction of path 1 and 2 in the occasion of incremental programming. That is to say, if path 1 advances along the negative direction of X axis, the value of the U is negative.

The range of the thread leading and the limit of spindle speed and so on are the same as that in G32 (Thread cutting). Thread chamfering can be finished in the thread cutting cycle and it is started by a signal from machine tool. Retracting distance r is set by the parameter #64 from 0.1L to 3.1L. (here L is leading).

The steps 1, 2, 3 and 4 can be finished by pressing the key of $\boxed{\text{cycle start}}$ again and again in single block mode.

（2）Application

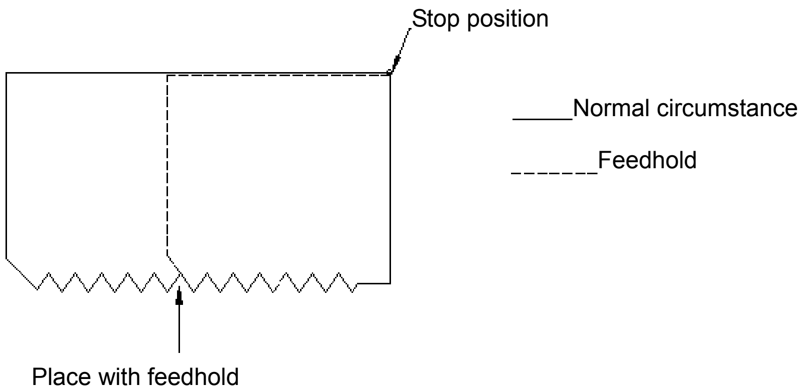
The application of thread chamfering avoids opening a clearance groove in the end of thread which the diameter is less than the bottom diameter. In this case, the strength of screw will be strengthened and it is very benefit for the screw need to be hermetic.



(a) Without chamfering (a) Chamfering

Note 1: The precautions are relative to the thread cutting, same as G32. The feedhold is supplied to achieve the dwell during the 2nd operation, and the tool will stop after completing the 3rd path at the rapid traverse rate.

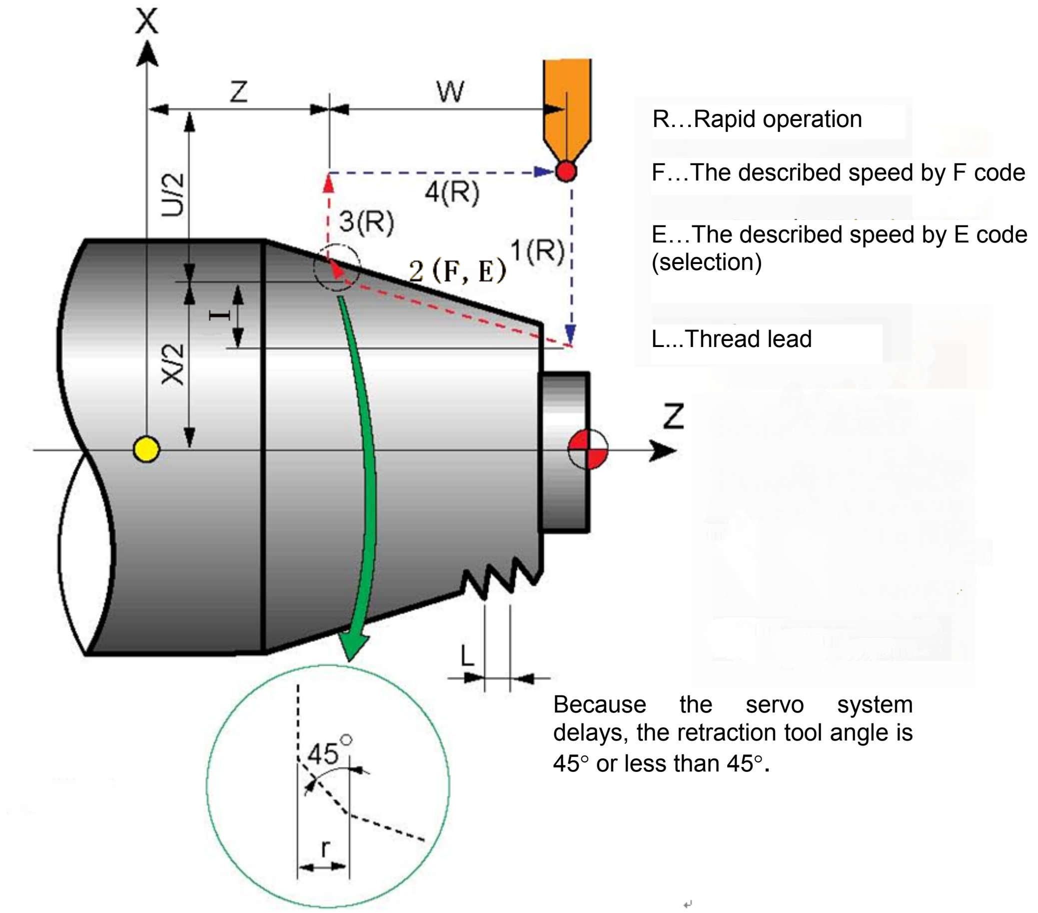
Note 2: When the dwell is applied in thread cutting, the tool will be retracted with diagonal as long as entering feedhold state in the 2nd step in thread cutting, and it returns to the start point along the X and Z axes separately.



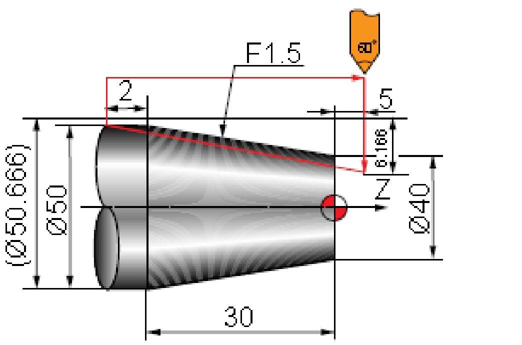
The others feedhold are invalid during retracting.

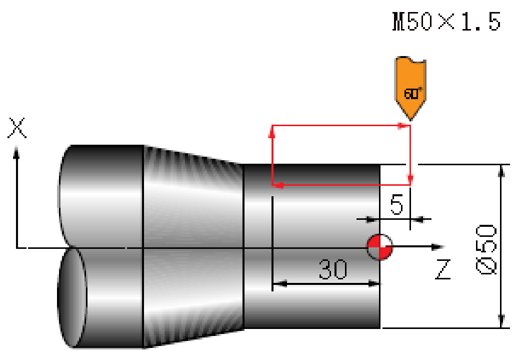
（3）Taper thread cutting command

$$\text{G92X（U）——Z（W）——I——}\left\{\begin{array}{l}\text{F——}\\ \text{E——}\end{array}\right\}\text{；}$$



Operation:





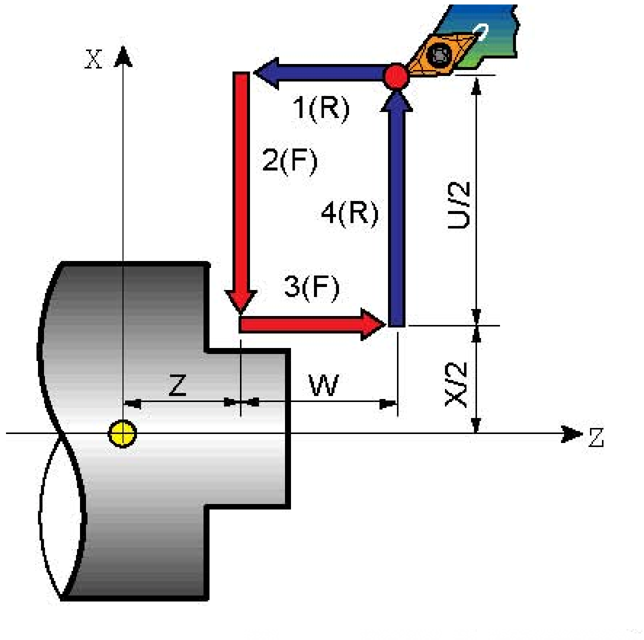

```
G30 U0 W0 ;
G50 S1000 T0100 ;
G97 S1000 M03 ;
G00 X70.0 Z5.0 T0101 M08 ;
G92 X49.4 Z－32.0 I－6.166 F1.5 ;
X49.0 ;
X48.7 ;
X48.5 ;
G30 U0 W0 ;
M30 ;
```

```
G30 U0 W0 ;
G50 S1000 T0100 ;
G97 S1000 M03 ;
G00 X60.0 Z5.0 T0101 M08 ;
G92 X49.5 Z－30.0 F1.5 ;
X49.2 ;
X48.9 ;
X48.7 ;
G30 U0 W0 ;
M30 ;
```

2.7.1.3　Inner, Outer End (Taper) Face Turning Cycle—— G94

（1）　End face machining uses the following command

G94X（U）——Z（W）——F——;



R…rapid traverse

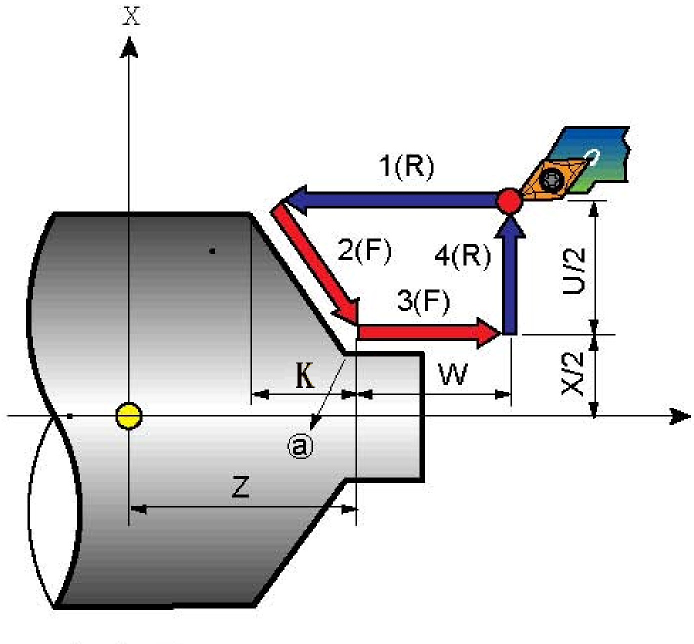
F…Described speed with F code

In the occasion of incremental programming, the positive/negative sign of digital followed address U and W is relative to the direction of path 1 and 2, That is to say, if path 1 advances along the negative direction of Z axis, the value of the W is negative.

The steps 1, 2, 3 and 4 can be finished by pressing the key of cycle start again and again in single block mode.

（2） Taper machining cycle is indicated by the following command

G94X（U）——Z（W）——K——F——；



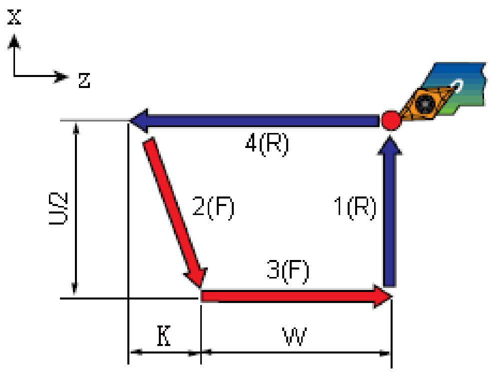
R…rapid traverse

F…Described speed with F code

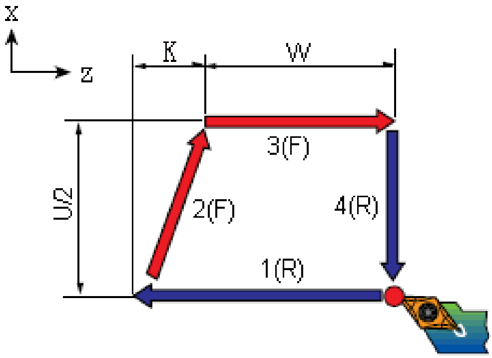
In the occasion of incremental programming, relationships between the positive/negative sign of

digital of address U, W and K and tool path is shown as follow:

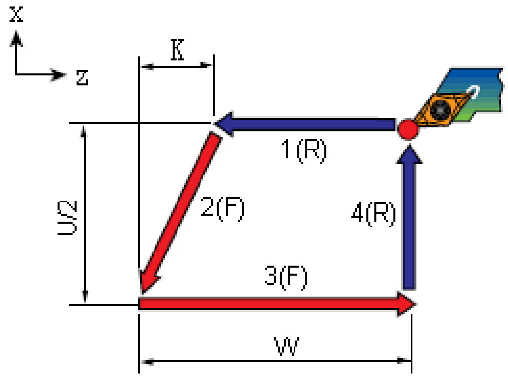
(i) U<0, W<0, K<0



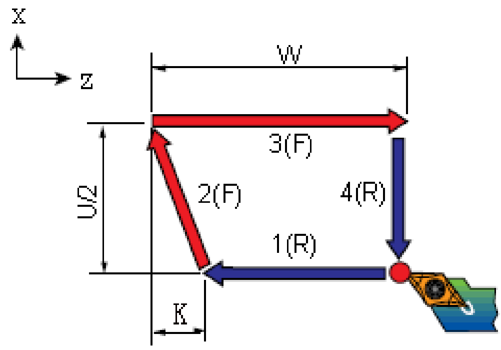
(ii) U >0, W<0, K<0



(iii) U<0, w<0, k>0 , |K| ≦ |W|



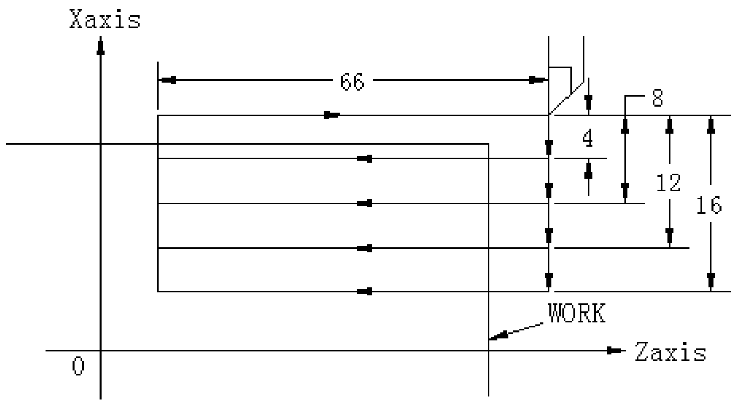
(iv) U>0, W<0, K<0 , |K| ≦ |W|



Note 1: The values of X (U), Z (W), I and K are modal sate during canned cycle, so if we are regardless of them, the values specified before are then valid.

So, when the Z axis displacement value is invariable, the displacement command canned cycle along with X axis can be repeatedly operated. See the following figure.

But, a one-shot G mode other than G04 is specified, or G code of group 01 other than G90, G92 and G94 are specified, and then the data is cleared.



The cycle in the above figure can be performed using the following programs

N030 G90 U-800 W-66000 F400;

N031 U-16000;

N032 U-24000;

N033 U-32000;

Note 2: The following two functions are effective by setting parameter #9.3(MOR).

(1) If the EOB (;) or the block including M30, M02 is executed after the canned cycle, the same cycle will be performed repeatedly.

(2) If M, S or T function is specified during canned cycle, the canned cycle and auxiliary function can be simultaneously performed, and if it is improper like this, the canned cycle (regulate G00 or G01) can be cancelled temporarily and then perform the M, S or T command. Canned cycle is performed again after M, S or T command.

For example: N003 T0101;

┆

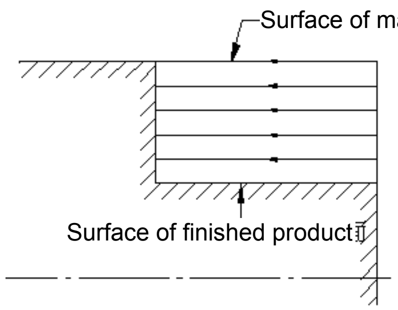
N010 G90 X20000 Z10000 F200；
N011 G00 T0202；
N012 G90 X20500 Z10000；

Note 3: The thread chamfering begins to perform as long as a signal from machine tool is received. But, the common machine tool builder deal with the signal like this: send it to machine tool with corresponding M code, this signal is used for thread chamfering and it is send to signal NC from machine tool.

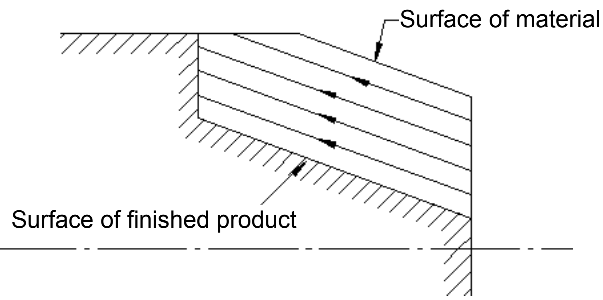
Note 4: When thread chamfering M code is applied in thread cutting, it must be specified in the block before G92, suppose that G92 is specified in the same block with this M code, the thread chamfering will not be performed because of the signal delay between machine tool and NC.

2.7.1.4 Canned Cycle Use

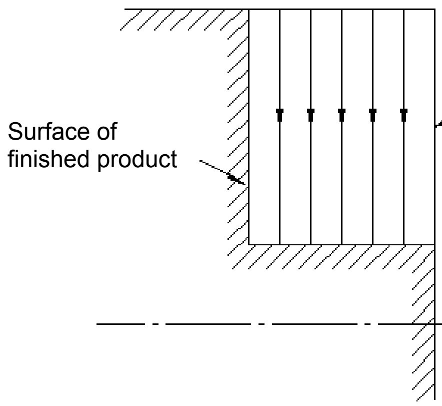
(1) Linear cutting cycle



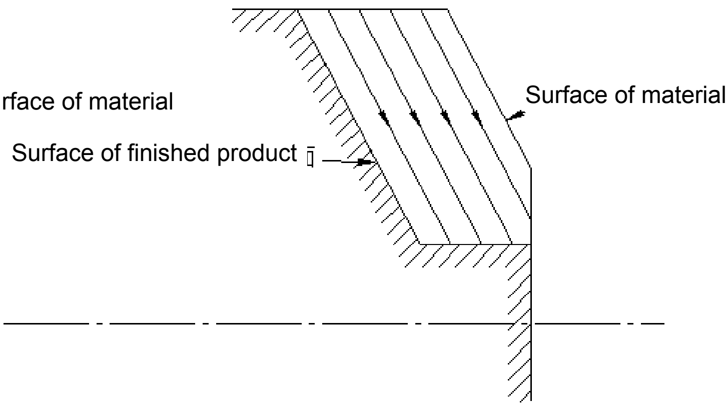
(2) Taper cutting cycle



(3) End face cutting cycle

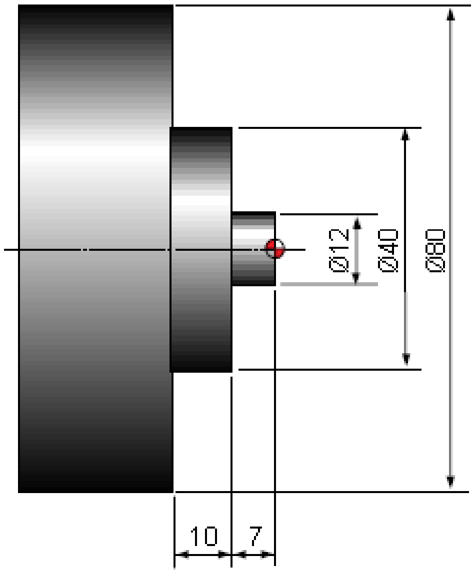


(4) Taper end face cutting cycle



Note: To protect against the overcut or the tool collision, the canned cycle start point should be set in the position beyond the range of closed figure formed by workpiece path.

Operation:



Example 1）

```
N10 G50 S2500 ;
G96 S180 M03 ;
T0300 ;
G00 X85.0 Z2.0 T0303 ;
G94 X12.0 Z-2.0 F200 ;
Z-4.0 ;
Z-6.0 ;
Z-7.0 ;
G00 X85.0 Z-5.0 ;
G94 X40.0 Z-9.0 F200 ;
Z-11.0 ;
Z-13.0 ;
Z-15.0 ;
Z-17.0 ;
G00 X200.0 Z200.0 T0300 ;
M30 ;
```

Example 2）

```
N10 G50 S2500 ;
G96 S180 M3 ;
T0300 ;
G0 X85.0 Z2.0 T0303 ;
G94 X12.0 Z-2.0 F200 ;
Z-4.0 ;
Z-6.0 ;
Z-7.0 ;
X 40.0 Z-9.0 ;
Z-11.0 ;
Z-13.0 ;
Z-15.0 ;
Z-17.0 ;
G0 X200.0 Z200.0 T0300 ;
M30 ;
```

2.7.2 Compound Canned Cycle

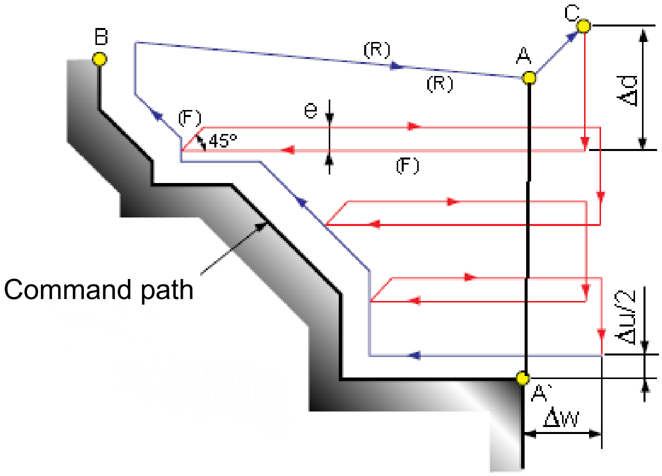
GSK983Ta's compound canned cycle includes: outer roughing cycle G71, end face cutting cycle G72, closed loop cutting cycle G73, finishing cycle G70, deep hole drilling cycle G74, outer grooving cycle G75 and thread cutting cycle G76. The system executes these codes and automatically counts cutting times and cutting path based on the programmed path, infeed amount and retraction amount, executes the cycle（tool infeed →cutting → tool retraction→tool infeed）many times, and automatically finishes roughing and finishing, and starting point and end point of its codes are the same.

2.7.2.1　Outer Roughing Cutting Cycle（G71）

G71 realizes a compound cycle roughing of a group of slant commands. It performs the axial cutting to the blank (called stepped roughing), and cuts along the path of command group, at last returns to the initial point and completes roughing.

There are two kinds of rough-turning cycle:

The first kind is as the following figure. If a program describes the finishing path from A-A'-B, the machine tool will turn away the specified part according to the cutting depth Δd, the rest of part of finishing is Δu/2 and Δw.



R…rapid traverse

F…feedrate

Retraction amount e is set by parameter #67(MRCDT) or parameter #67 in SETTING page. It is parallel to the Z axis to cut with Δd depth at the beginning, and then, it is parallel to the programmed path cutting.

This command describes as follows:

G71P（ns）Q（nf）U（Δu）W（Δw）D（Δd）F（Δf）S（s）T（t）;

N（ns）……

F——

S——

T——

The movement command from A-A'-B is specified from sequence number ns to nf in the block.

N（nf）……

P: The first sequence number of the program of path A-B. (ns)

Q: The last sequence number of the program of path A-B. (nf)

U: Diameter programming is performed along with the distance of X axis remainder and direction.

W: Distance and direction along with Z axis remainder

D: Tool feed value Δd is not described for the sign.

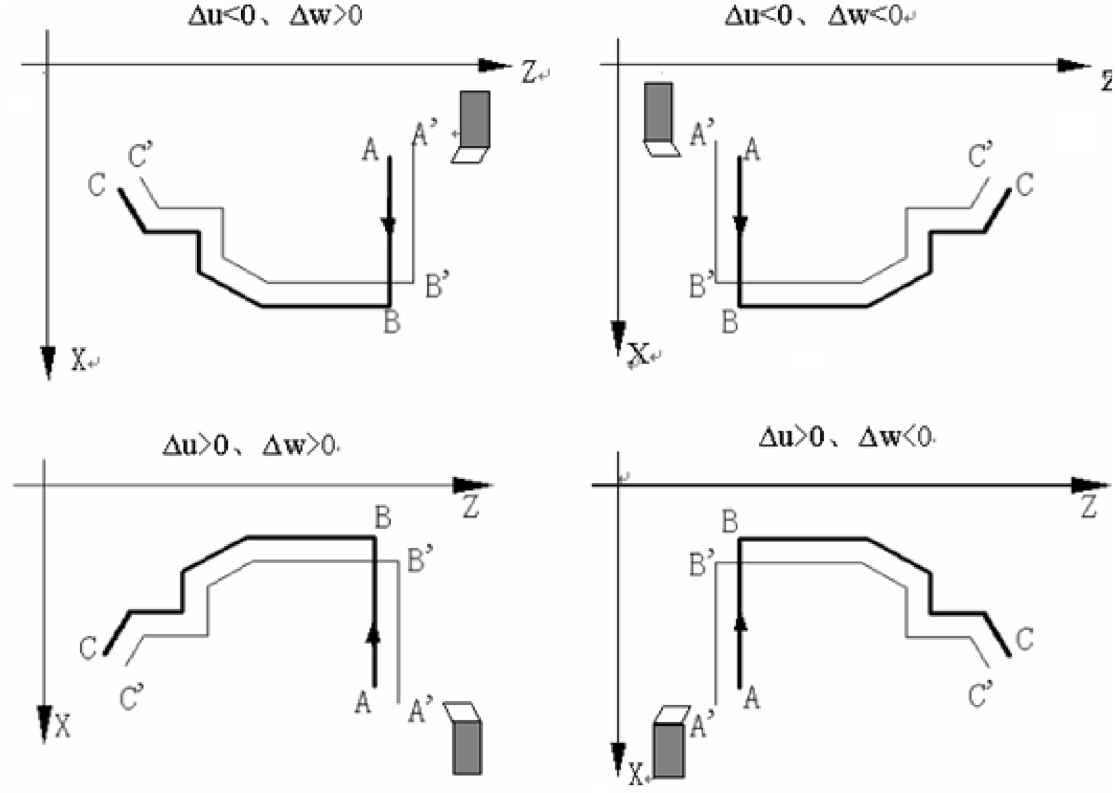
Tool feeds according to the direction of A-A'.

F, S, T: All the functions of F, S, T included in the canned cycle program ns to nf are ignored when the G71 is performed. But, it is effective when G70 executes.

All the F, S, T function specified in G71 block or before G71 block are enabled. If the surface

constant speed control function is selected, the command G96 or G97 between ns and nf is disabled, but, it is effective if it is specified in G71 block or before G71 block.

Note that the following four kinds of cutting mode. All of these cutting cycles are performed to parallel to the Z axis, the signs of Δu and Δw are as follows. B→C is the finishing path, B'→C' is the roughing contour and A is the start tool point.



Tool path from A—B is specified in the sequence number ns in a block, the movement command along Z axis can not specified in this program. The tool path from B-C must be up or down monotonously.

When G00/G01 is applied in the tool path program A-A', the cutting or positioning should be separately finished in G00/G01 mode.

If the tool nose center is regarded as machining start which it is completed the machining with the command of cutter compensation G71. The tool radius offset value is added to the Δu and Δw, as for the roughing, the tool nose radius is O.

Δu=Δu + tool nose radius offset value

Δw=Δw + tool nose radius offset value

Note 1: The block between the sequence number ns and nf can not call the subprogram.

Note 2: The rest of finishing allowance of the workpiece figure cutting is performed at the last of canned cycle, but, parameter #307.7 can be set for cancelling this step.

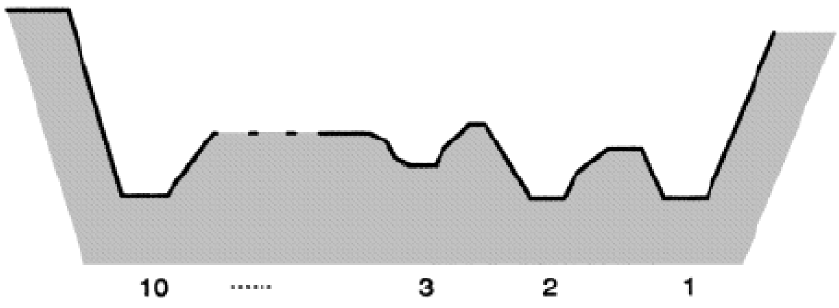
Note 3: In the case of note 2, due to a surface finishing program is stopped from ns to nf, and a linear interpolation G01 is not always arrives the end (G02 or G03 can be done it)

Note 4: Sine T function is omitted in G71 block and it is valid in G70 block, attention should be paid that whether the T code is required in programming, if not, cancel it.

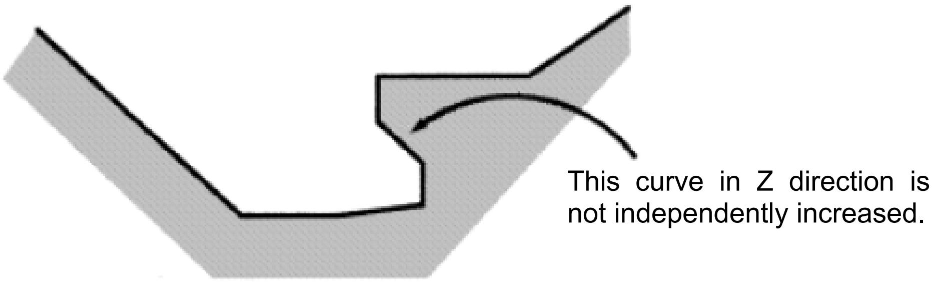
Note 5: Z(w) command can not be included in the block with the sequence number ns.

（1） The second type is different from the first type:

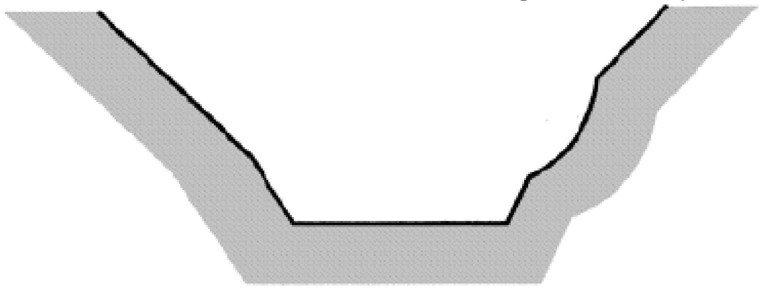
（I） The monotone increasing or monotone decreasing for the workpiece figure along X axis is not required. Allowing for up to 10 grooves:



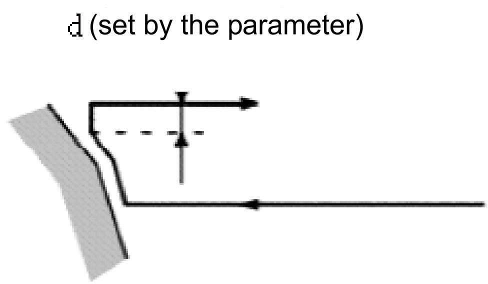
But, the curve in the Z direction must independent, the following curve is incorrect:



(II) The first cutting of roughing is not required for vertical to Z direction cutting. Any figure, the Z direction can be machined as long as it is independently increased:

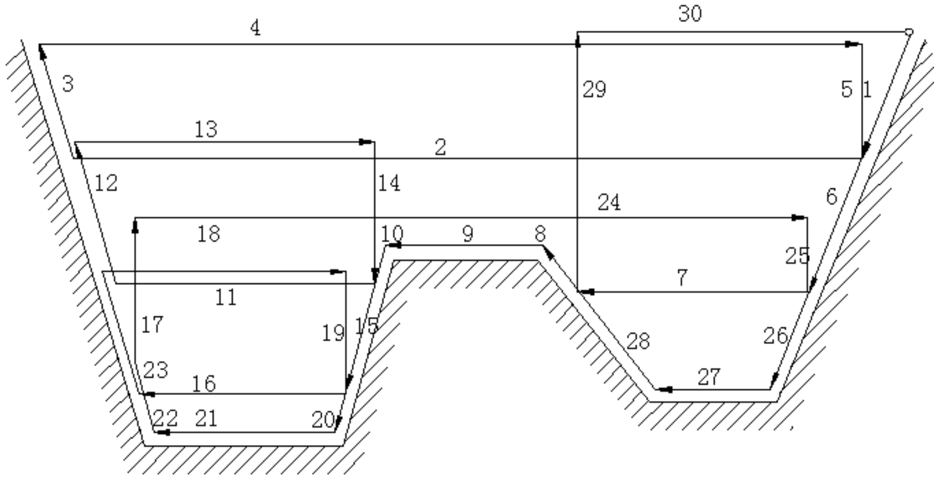


(III) A tool retraction can be applied after cutting along with the workpiece surface.



The tool retraction amount d regarded as parameter #67 must be set in advance.

(IV) The cutting path with two slots is shown below.



(V) The tool nose compensation value may not add to finishing allowance Δu and Δw, and the tool nose radius offset is regarded as 0.

(VI) Normally, the Δw must be set to 0, or, the workpiece overcut may occur and the tool nose will cut into the side of the workpiece.

(VII) X (U) and Z (W) must be indicated in the first block of specifying workpiece form. If the Z axis movement is not applied in the first block, the W0 is then specified.

(2) Usage for two kinds of cycles

First kind : Specify an axis in the first block of specifying workpiece form.

Second kind : Specify two axes in the first block of specifying workpiece form.

When the Z axis movement is not applied in the first kind, and machining with the second kind, W should be set to 0.

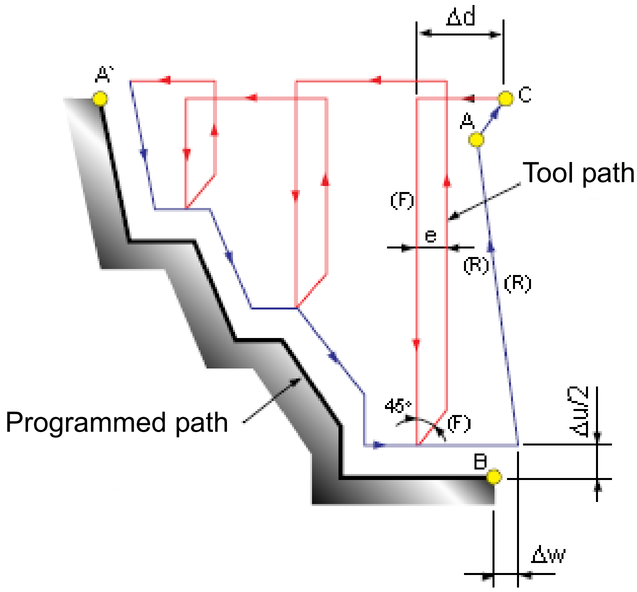
For example:

The first kind	the second kind
G71 P100 Q200……	G71 P100 Q200……；
N100 X（U）　；	N100X（U）Z（W）；
┆	┆
N200………………；	N200………………；

2.7.2.2　End Face Roughing Cycle（G72）

G72P <u>(ns)</u> Q <u>(nf)</u> U <u>(Δu)</u> W <u>(Δw)</u> D <u>(Δd)</u> F <u>(f)</u> S <u>(s)</u> T <u>(t)</u>

The execution of this cycle is the same as G71 other than parallel to X axis, it is shown as below:



R…rapid traverse

F…feedrate

Tool retraction amount is set by parameter #67 (MRCDT), or by parameter #067 in SETTING page.

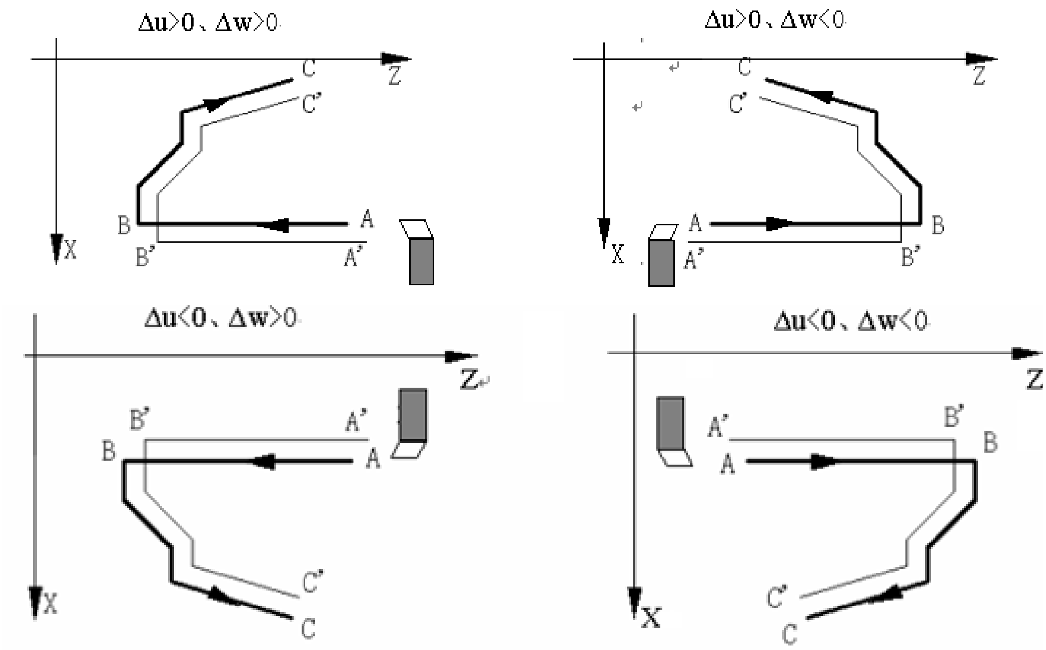
Explanation:

1) Cycle cutting parallel with X axis is done.

2) **Coordinate offset direction with finishing allowance:** Δu, Δw indicate coordinate offset and cut-in direction when finishing. Sign relationship of Δu, Δw are shown below. In the figure: B→C is the finishing path, B'→C' is the roughing contour, and point A is the start tool point.

There are four kinds of machined path by executing G72. These paths can be machined by the repetition operation parallel to X axis.

Signs of Δu and Δw are as follows:



The tool path machining mode from A to B (cutting feed or position) is determined by which either G00 or G01 is specified in this block. It is same to the G71.

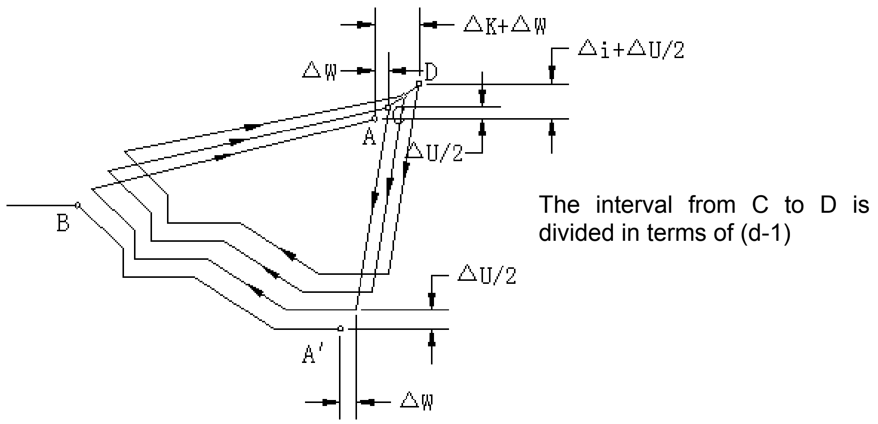
The tool path from B to C must be specified in the first block (sequence number is ns), but the movement along Z axis can not be indicated in this block. The tool path from A' to B must be monotonously increased/decreased.

If the machining start is programmed with tool nose center, and also, it completed the machining with tool nose center compensation G72 command, the tool nose center compensation value must be added to Δu and Δw, it is the same as that in G71.

Note 1: The terminal face rough cutting cycle tool path must be monotonously increased/decreased.
Note 2: Z(W) command can not be included in the block with sequence number ns.

2.7.2.3 Closed Loop Cutting Cycle（G73）

This cycle is repeatedly cut with canned figure and moving the tool position step by step, so that the blank can be machined with high efficiency by this cycle, such as forging and casting.



Defining the figure A-A'-B to the program.

The program commands are as follows:

G73 P（ns）Q（nf）I（Δi）K（Δk）U（Δu）W（Δw）D（d）F（f）S（s）;

N（ns）……;
┆ The commands from A-A'-B are indicated into the block which its sequence number is from ns to nf.
N（nf）……;

P: The first sequence number of part machining block.

Q: The last sequence number of this part is machined in the block.

I: Retraction distance and direction along with X axis.

（Δi）……Radius programming

K: Retraction distance and direction along with Z axis.

（ΔK）

U: The finishing remainder with respect to X axis direction.

（Δu）……Diameter programming

W: The finishing remainder with respect to Z axis.

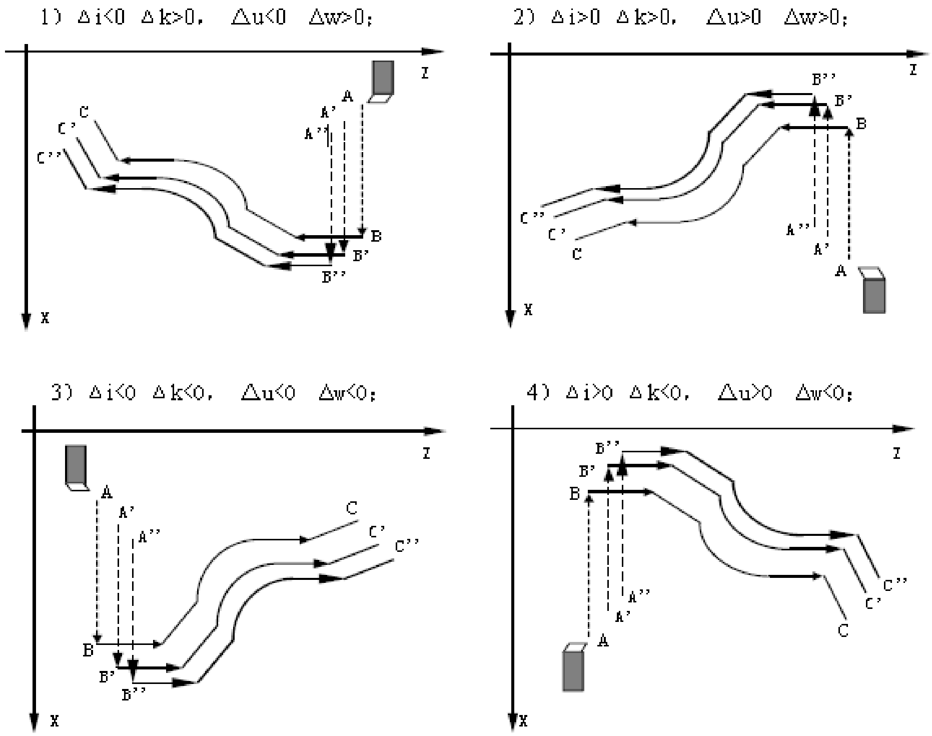
（Δw）

D: Segmentation number

(Δd)……this value is equal to the repetition times of roughing.

F, S, T: The F, S and T functions in G73 block are is valid, but the F, S and T functions in the block in which the sequence number from ns to nf are all ignored.

According to the different signs of Δu, Δw, Δi and Δk, there are four kinds of different feed mode by G73 cycle machining. A is the start tool point, B→C is the workpiece contour, B'→C' is the roughing contour, B"→C" is the finishing path. The tool returns to point A after the cycle is completed.



If the machining start is programmed with tool nose center, and also, it is completed the machining with cutter compensation G73 command, the cutter compensation value must be added to Δu and Δw, same as G71.

2.7.2.4　Finishing Cycle（G70）

The tool executes finishing from starting point along the blocks ns～nf specified by the workpiece finishing machining path. After G71, G72, G73 executes roughing, G70 does finishing, and completes the finishing allowance one time. After G70 is completed, the tool returns to the starting point and executes the next block following G70.

Roughing uses G71, G72 and G73 command, and finishing uses G70.

G70p（ns）Q(nf);

P: The sequence number of the first block uses workpiece finishing.

Q: The sequence number of the last block uses workpiece finishing.

G70 path is determined by programmed path of ns～nf blocks. Relationship between ns and nf in G70～G73 is shown below:

```
．．．．．．．．
G71/G72/G73 ……；
N__(ns) ．．．．．．
．．．．．．．．
      ·F
      ·S                    Block group for finishing path
      ·
      ·
N__(nf)……
      ．．．
G70 P(ns) Q(nf)；
      ．．．
```

Note 1: F, S and T functions specified in the G71, G72 and G73 blocks are invalid, but, in G70 mode, they are valid in the block from ns to nf.

Note 2: The compound canned cycle (G71, G72 and G73) should be completed in the Auto mode.

Note 3: When the roughing is performed with G71, G72 and G73, up to 3 programs determined by address P and Q can be stored. So, when G70 executes, search is not needed. The block determined by address P and Q can be rapidly found out. After several roughing (G71, G72 and G73) are finished, and several fine-machining cycle can be performed together. In this case, the finishing as for the 4th cycle and its followed will be delayed to execute, because the memory searches.

G71 P100 Q200……；

N100……

………

………

N200

G71 P300 Q400……；

N300……

………

………

N400

………

………

G70P100Q200； **The finish-machining can be performed for each time, and the search is not performed when it is not more than 3 cycles**;

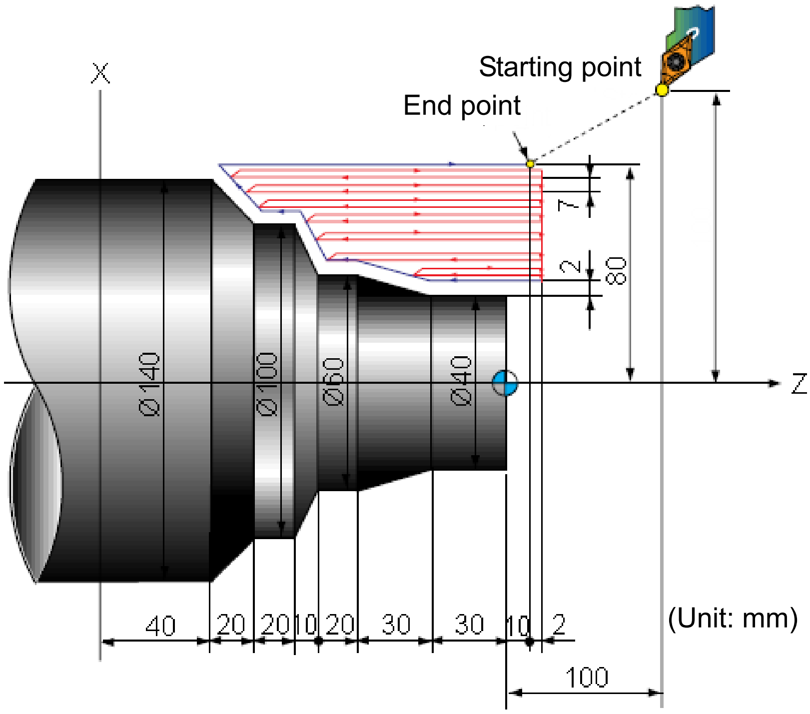
G70P300Q400； **The search is performed when it is more than 4 cycles.**

Note 4: Tool returns to the start point at the rapid traverse rate when G70 command is completed. The NC data followed by G70 command in the block is read into.

Note 5: For composite canned value cycle (G70—G73), the subprogram can not be called in the block of sequence numbers are from ns to nf.

Note 6: The memory address is stored with G71, G72 and G73 during rough-lathe cycle. Cancel is performed after the completion of G70. The memory address stored can be erased by resetting operations.

Example 7, 2, 1: Composite cycle programming (G70, G71)



（Metric and diameter programming）

N010　G50 X200.0 Z220.0；

N011　G00 X160.0 Z180.0；

N012　G71 P013 Q019 U4.0 W2.0 D7000 F30 S55；

N013　G00 X40.0 F15S58；

N014　G01W-40.0；

N015　X60.0W-30.0；

N016　W-20.0；

N017　X100.0W-10.0；

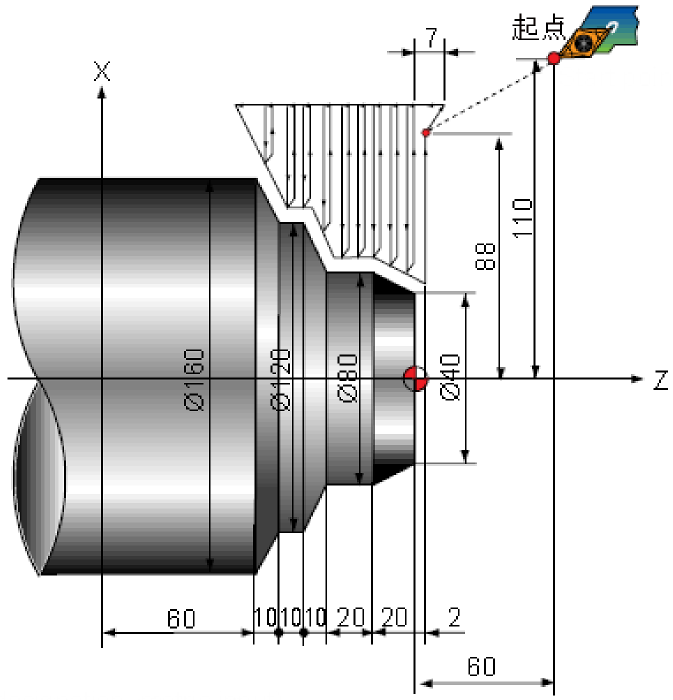
N018　W-20.0；

N019　X140.0W-20.0；

N020　G70 P013 Q019；

Example: 7.2.2 composite canned cycle programming (G70, G72)

Starting point



（Diameter and metric programming）

N010　G50 X220.0 Z190.0

N011　G00 X176.0 Z132.0；

N012　G72 P013 Q018 U4.0 W2.0 D7000 F30 S55；

N013　G00 Z58.0 F15 S58；

N014　G01 X120.0 W12.0；

N015　W10.0；

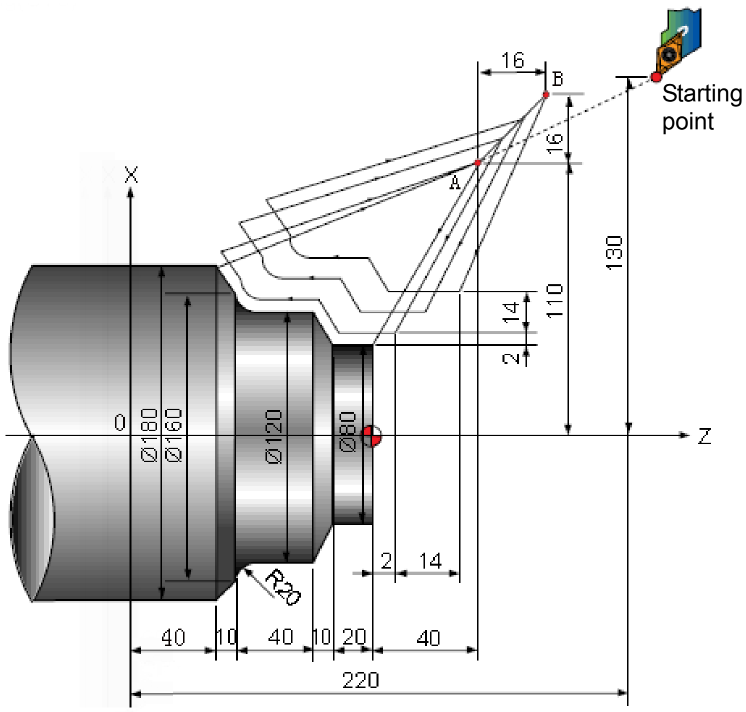
N016　X80.0 W10.0

N017　W20.0；

N018　X36.0 W22.0

N019　G70 P013 Q018；

Example: 7.2.3 composite canned cycle programming (G70, G73)



（Diameter and metric programming）

N010　G50 X260.0 Z220.0；

N011　G00 X220.0 Z160.0；

N012　G73 P013 Q018 I14.0 K14.0 U4.0 W2.0 D3 F30 S0 180；

N013　G00 X80.0 W-40.0；

N014　G01 W-20.0 F10.0；

N015　X120.0 W-10.0；

N016　W-20.0 S0400；

N017　G02 X160.0 W-20.0 120.0；

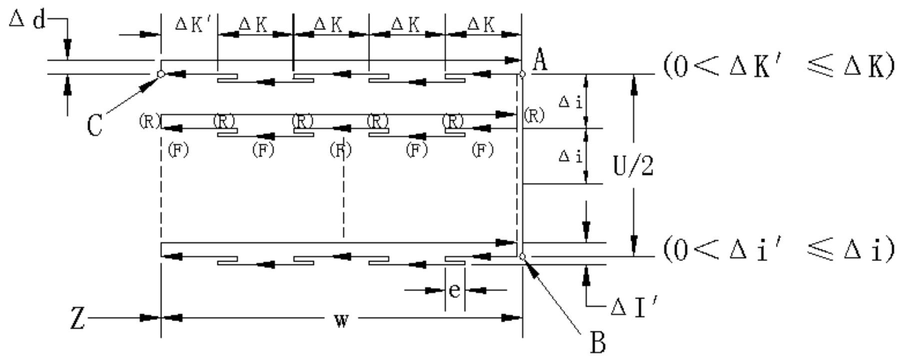
N018　G01 X180.0 W-10.0 S0280；

N019 G70 P013 Q018；

2.7.2.5　Deep Hole Drilling Cycle（G74）

Execute the axial (Z axis) feed, retraction, and feed again from starting point till the end point of Z axis, and execute the radial retraction, axial retraction to starting point of Z axis, and one-time axial cutting cycle is completed; after the radial tool infeed is done, the next axial cutting cycle is executed; after cutting to end point is done, the tool returns to starting point (starting point and end point of G74 are the same), and the axial grooving cycle is completed. Directions of radial tool infeed and axial tool infeed in G74 are determined by cutting point X(U), Z(W) and starting point. The code is used to machine the ring groove or center deep hole, inconsecutive axial cutting to break chip, and time remove chip on the workpiece surface.

G74 X（U）___Z（W）___I（Δi）K(Δk) F___ D(△d)；



X: X component of point B

U: Incremental from A to B

Z: Z component of point C

W: Incremental from A to C

I: The movement value along with X direction (without sign) (Δi)

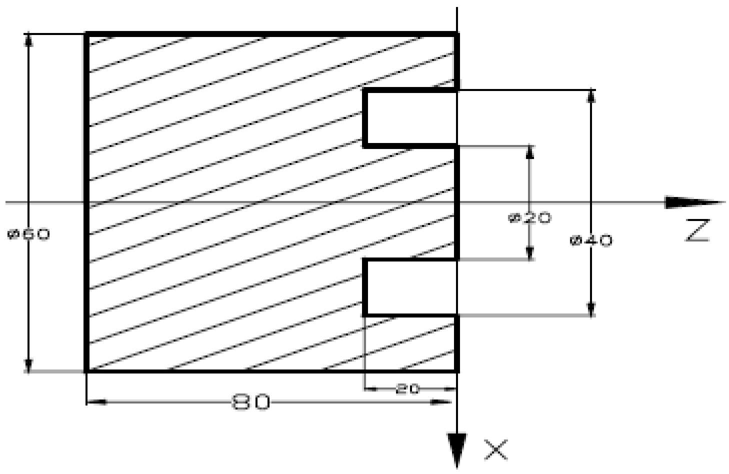
K: Cutting value along with Z direction (without sign) (Δk)

D: The retracting value at the bottom of hole (Δd)

(When the D is not applied, it is regarded as zero.)

F: feedrate

Note: The retraction value e in cutting can be set with parameter #65 (GROVE), or set in parameter #65 (GROVE) in SETTING page.



Program(suppose the grooving tool width is 4mm, and the least increment is 0.001mm):

O0007；

G0 X36.0 Z5.0 M3 S500; **(Start the spindle and position to the machining starting point, add the tool width in X direction)**

G74 D0.5； **(Machining cycle)**

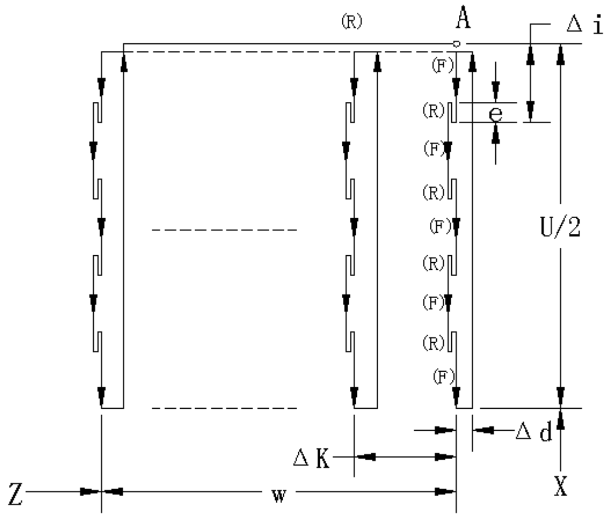
G74 X20.0 Z-20.0 I3000 K5000 F50；(Tool infeed 5mm in Z direction, tool retraction 0.5mm，After infeed to end point (Z-20), rapidly return to starting point (Z5)，tool infeed 3mm in X direction. Execute the above cycle to continuously run)

M30； **(End of program)**

2.7.2.6 Outer Grooving Cycle（G75）

The following command in the figure is used to finish the cycle. This case is the same as G74 except the Z axis replaced X axis. Its character is drilling the depth hole along with X axis and easy to chip-removal and outer lathe slot along with X axis. (Z, W and K must be omitted in this case)

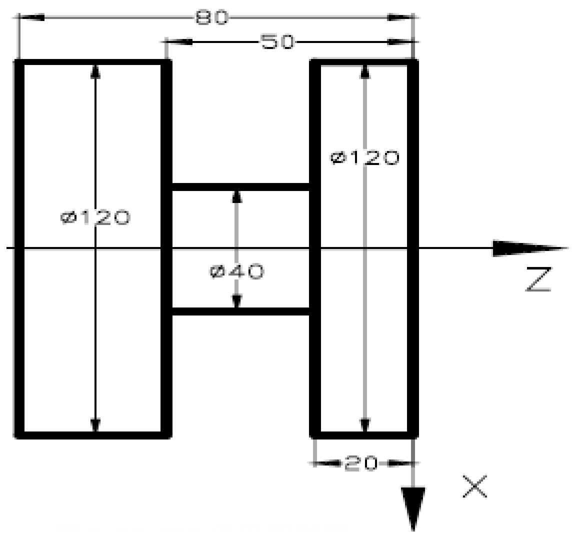
G75X（U）___Z（W）___I（Δi）K(Δk)F___D(Δd)；



G74 and G75 are used for slotting and drilling separately, and they can be retracted automatically, there are four kinds of feed direction for them.

Note: The retraction value e in cutting can be set with parameter #65 (GROVE), or set in parameter #65 (GROVE) in SETTING page.

Operation:



Program(suppose the grooving tool width is 4mm, and the least increment is 0.001mm):

O0008；

G00 X150.0 Z50.0 M3 S500； **(Start the spindle, speed 500)**

G0 X125.0 Z-24.0； **(Position to machining starting point, add the tool width in Z direction)**

G75 D0.5 F150； **(Machining cycle)**

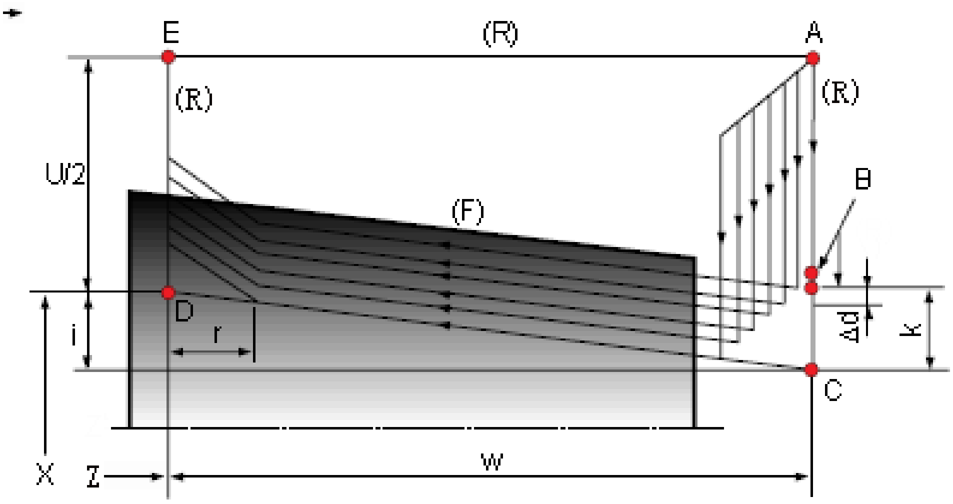
G75 X40 Z-50 I6000 K3000； **(Tool infeed 6mm every time, tool retraction 0.5mm, after feedrate to end point(X40), rapidly return to starting point (X125), tool infeed 3mm in Z direction, execute the above cycle to continuously run)**

G0 X150.0 Z50.0； **(Return to machining starting point)**

M30； **(End of program)**

2.7.2.7　Thread Cutting（G76）

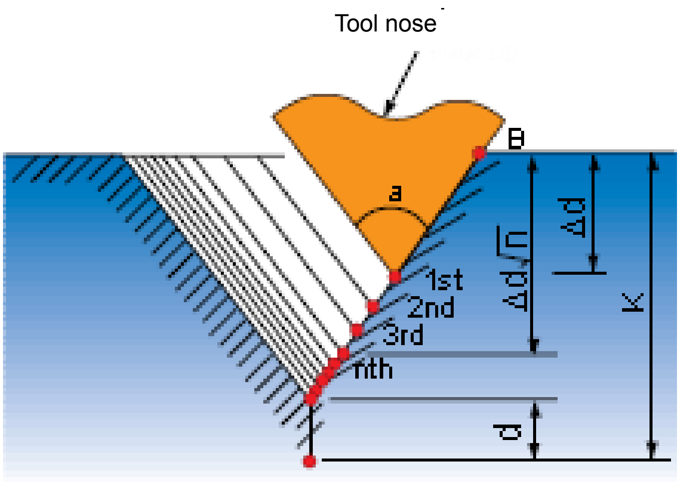
The following figure indicate thread cutting cycle, it is programmed with G76 command.



R…Rapid traverse

F…Feedrate

Note：When Ⅰ ＜ IWI, there is no need to add retraction flute; When Ⅰ ≥ IWI, retraction flute should be added.



G76 X(U) __Z(W) __I__ K__D(Δd) { F__ / E__ } A__ ；

I: Radius error (i) of thread
If I=0, the common straight thread can be cut.

K: Thread height (X axis direction) (K) is specified with radius value.

D: The first cutting depth (△d)

F, E: Thread leading (Same as G32)

A: Tool nose angle (thread angle). There are six angles can be selected:

It is A80 in the occasion of 80°,

It is A60 in the occasion of 60°,

It is A55 in the occasion of 55°,

It is A30 in the occasion of 30°,

It is A29 in the occasion of 29°,

It is A0 in the occasion of 0°,

If an A is omitted, 0 by default.

R：Thread chamfering

When the thread leading is indicated with L, the r value is in the range of 01~12.7L， 0.1Lis regarded as incremental and it can be set by parameter #64 (THDCH).

Tool nose loading can be released in this cycle by cutting with cutting edge. If the first cutting value is △d, then the Nth cutting value is $\triangle d\sqrt{n}$, each cutting value is stated. The least cutting depth can be operated #68 (THCLM) with parameter SETTING. There are four kinds of feed direction, also, the inner thread can be machined.

The thread cutting is shown as the above figure, the cutting feedrate between C and D are described with F or E code. The tool in the other path is rapid feedrate.

The incremental signs in the above cycle are shown below:

U, W: Negative (It is determined by the directions from A to C and from C to D)

I: Negative (It is determined by the direction from A to C)

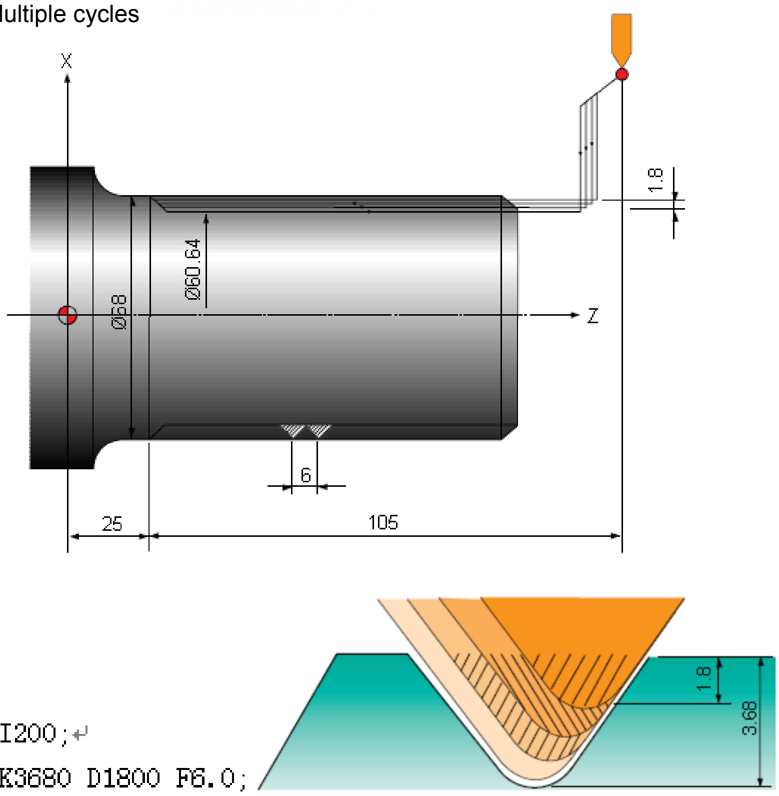
K: Positive (always positive)

D: Positive (always positive)

The finish-lathe remainder value α (Radius programming) can be set by parameter #66 (THDFN), the range is indicated as 0.000～16.383 mm or 0.000～1.6383 inch.

Example 7.2.7.1: Multiple cycle program (G76)

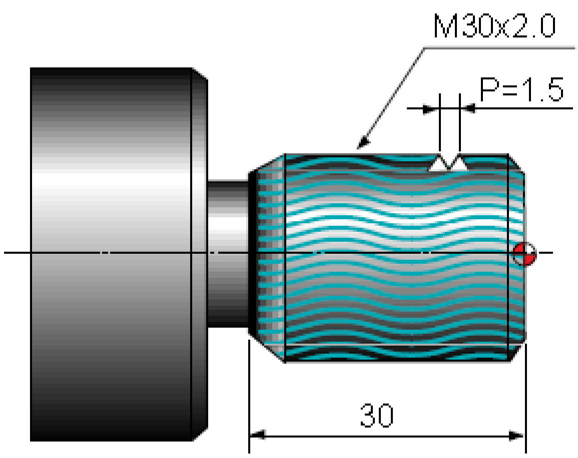
Multiple cycles




```
G00 X80.0 Z130.0;
G76 K011060 D100 I200;
G76 X60.64 Z25.0 K3680 D1800 F6.0;
```

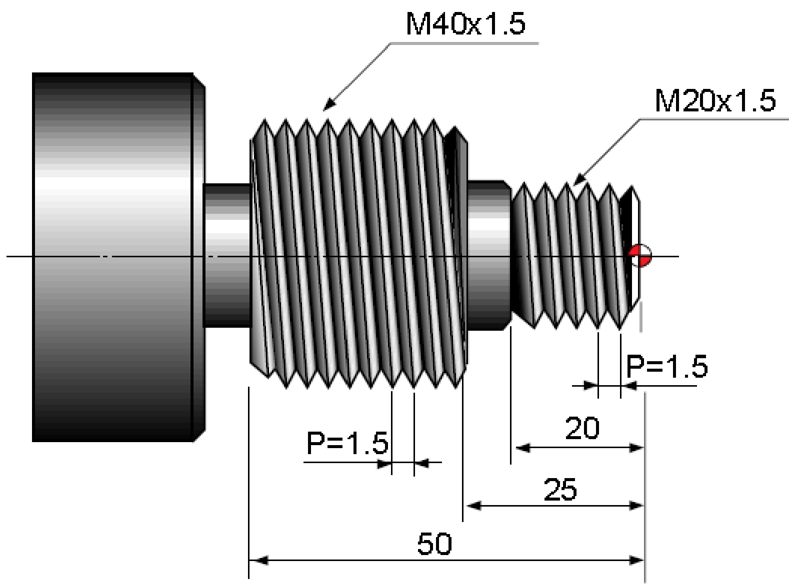
Note：The precaution of thread cutting is same to the G32 and G92 thread cutting cycle. When entering keep state during thread cutting, the dwell is effective during thread cutting, the tool retracts to the cycle start point.

Operation 1：




```
N10 G97 S1000 M03;
T0100;
G00 X50.0 Z5.0 T0101;
G76 K021060 D100 I100;
G76 X28.2 Z-32.0 K900 D500 F1.5;
G00 X200.0 Z200.0 T0100;
M30;
```

Operation 2:



N10 G97 S800 M03;
T0300;
G00 X30.0 Z5.0 T0303;
G76 K021060 D100 I100;
G76 X18.2 Z-20.0 K900 D500 F1.5;
G00 X50.0 Z-20.0;
G76 K021060 D100 I100;
Can be omitted
G76 X38.2 Z-52.0 K900 D500 F1.5;
G00 X200.0 Z200.0 T0300;
M30;

2.7.2.8 Notes for Compound Canned Cycle (G70~G76)

(1) The address P, Q, X, Z, U, W, I, K, D and A should be correctly specified in the block of composite canned cycle. Set correct address and its value, not to lack the address or set some address data beyond the permission range in programming.

(2) When G71, G72 or G73 is specified, G code (G00 or G01) of group 01 must be indicated in the block of the sequence specified with P, or No.065 alarm occurs.

(3) G70, G71, G72 and G73 can not be instructed in MDI mode, or, No.067 alarm occurs. However, the G74, G75 or G76 can be instructed in MDI and Auto modes.

(4) M98 and M99 can not specified in the block which it is included G70, G71, G72 or G73 and between the blocks which is specified by P and Q.

(5) For the occasion of G70, G71, G72 and G73, the following commands can not be applied between the blocks which are specified with P and Q.

★ The one-shot G code other than G04 (Dwell)

★ G code of group 01 other than G00, G01, G02 and G03

★ G codes of group 05 and 06

★ M98/M99

(6) This dwell is valid and manual operation can be inserted when the canned cycle is performed. But, this cycle should be returned and restarted at the beginning of manual operation.

When the manual operation is performed, if the tool not retracts to its original position when restarting again, even if the manual switches are opened, the tool offsets the tool path because of the movement amount of manual operation.

(7) The sequence number between block P and Q must not same to the one of the stored block in program, when the G70, G71, G72 or G73 is executing.

(8) In G70, G71, G72 and G73, chamfering or a corner R function can not be used in the block specified with P, Q, or, No.069 alarm occurs.

（9）In G71~G76 cutting cycle, the tool nose radius compensation is no used.

（10）Marco program interruption function can not be used when compound cycle is being executed.

（11）In general condition, the tool start point should be set in the position beyond the range of closed figure formed by the workpiece path; And, the end coordinate of the block between P and Q should be correct, check the program and the machining path before workpiece processing to protect against the overcut or tool collision.

2.8 Spindle Speed Function (S Function), Tool Function (T Function), Miscellaneous Function (M Function)

The BCD code and strobe signal are sent to NC system by the digitals specified after S, T and M. These signals are mainly used for controlling the machine switch function.

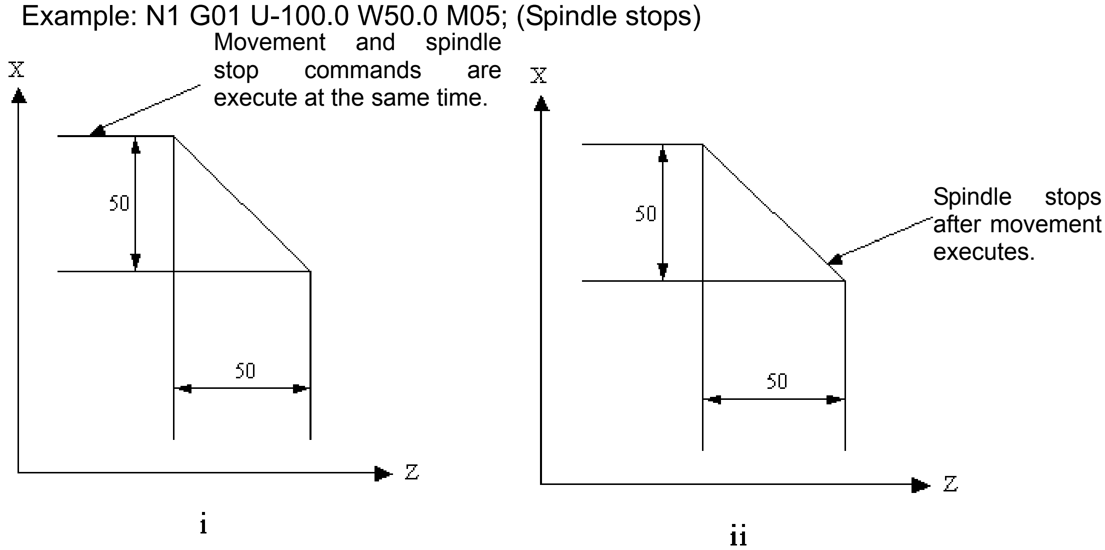
S code is used for spindle control, T code is used for tool change, and M code is used for kinds of machine switch functions. Since function configuration of the machine tool builders not always the same, refer to the user manual issued by the machine builder about the address and code.

When a movement command and S, T, M codes are specified at the same block, there are two ways to perform the following commands.

(1) The movement command and S, T, M codes are performed simultaneously.

(2) S, T, M codes are performed after the movement command is executed.

Example: N1 G01 U-100.0 W50.0 M05; (Spindle stops)



(Diameter programming)

One of method from above is selected in terms of technical requirement of machine builder. refer to the manual issued by the machine tool builder for details.

2.8.1 Spindle Speed Function (S Function)

2.8.1.1 S2 Digital

Address S and its following 2 digits are controlled to the spindle speed; refer to the manual issued by the machine tool builder for details.

Note: When the 4 digits are specified in the code of 2 digits, the following two digits are effective.

2.8.1.2 S4 Digital

Spindle speed (rpm) is directly determined by the S and the following 4 digits of S.

Spindle speed unit depends on the different machine builders.

2.8.2 Constant Surface Cutting Speed Control

2.8.2.1 Constant Surface Cutting Speed

The surface speed spindle revolution is specified after S, it is regardless of tool position when the spindle turns but the surface speed keeps constant. The voltage backfeed to the spindle control, so, spindle revolution generates the correct surface speed.

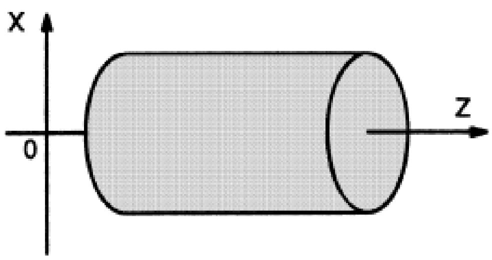
Surface speed unit is as follows:

Input unit	Surface speed unit
mm	m/min.
Inches	Feet/min

The speed unit is determined by the machine tool builder. The following G code is used for constant surface speed control.

G code	Definition	Unit
G96	Constant surface cutting speed control	m/min. feed/min.
G97	Specifying spindle speed	r/min

When the surface speed control is effective, the workpiece coordinate system should be set to the Z axis becomes revolving axis (X=0).



The S in the mode of G96 is regarded as S=0, when M03 and M04 are not specified. The S is enabled when M03 or M04 is specified.

2.8.2.2 Spindle Speed Override

The specified signal is transformed from machine tool which can be set the specified surface speed or spindle feedrate into eight control shifts, such as 50, 60, 70, 80, 90, 100, 110 and 120%.

Note: The parameter＃9.5（SOV）should be set to 1 to make the spindle speed override valid. When the SOV is set to 0, the spindle speed override is invalid.

2.8.2.3 Max. Spindle Speed Clamping

The max. spindle speed is specified in G50 S (the constant surface speed unit is r/min)

G50 S__；

When the spindle speed controlled on the constant surface speed is reached to the above value which is specified with program, this speed is clamped at the max. value.

When a constant surface speed control is selected, and the maximum spindle speed can be set with parameter＃134 (SPDMAX). In this case, when the constant surface speed control mode (G96) or (G97) is applied, the spindle speed is limited below the max. speed.

2.8.2.4 Rapid Traverse（G00）

In the rapid traverse block included G00, the cutting will not be performed in rapid traverse, then the constant surface cutting speed is not changed along with the changes of tool position, and the corresponding constant surface speed will be calculated at the program end position and sent to the spindle.

Note 1: When parameter ＃9.4（SSCR）is set to 0, the surface speed should be calculated in G00 mode along with tool position.

Note 2: The max. spindle speed is without set when the power is turned on, the speed is not clamped.

Note 3: If the max. spindle speed is set with G50 command, the clamping is effective to G96 command only, not for G97. If the max. spindle speed is set with parameter＃134（SPDMAX）, the clamping is then effective for G96 or G97.

Note 4: G50 S0; It indicates that the spindle speed is clamped at 0rpm.

Note 5: The S value specified in G96 mode is also used for G97 mode, and it is restored when turning to G96.

G96 S50；	**(50m/min. or 50feet/min)**
G96 S50;	**(50m/min. or 50feed/min)**
G97 S1000；	**(1000rpm)**
G97 S1000;	**(1000rpm)**
G96 X1000；	**(50 m/min. or 50 feet/min)**
G96 X1000;	**(50 m/min. or 50 feed/min)**

Note 6: The surface speed in the constant surface speed control is only suited to the programmed path, but it is not suited for the tool path after offset.

Note 7: Even if the machine is not run in the state of machine locking, the coordinate value constant surface speed of X axis is computed to the corresponding changeable program.

Note 8: In thread cutting mode, the constant surface speed is effective. Since the spindle speed responses to the NC system, it is better to cancel the constant surface speed control in taper thread cutting.

Note 9: G96 and G98 can be valid at the same time.

Note 10: When G96 is turned to G97 mode, if the S (rpm) is not specified in G97 block, the spindle speed is

lastly specified with G96 is regarded as the S value in G97.

N111 G97 S800;　　　800 rpm

⋮

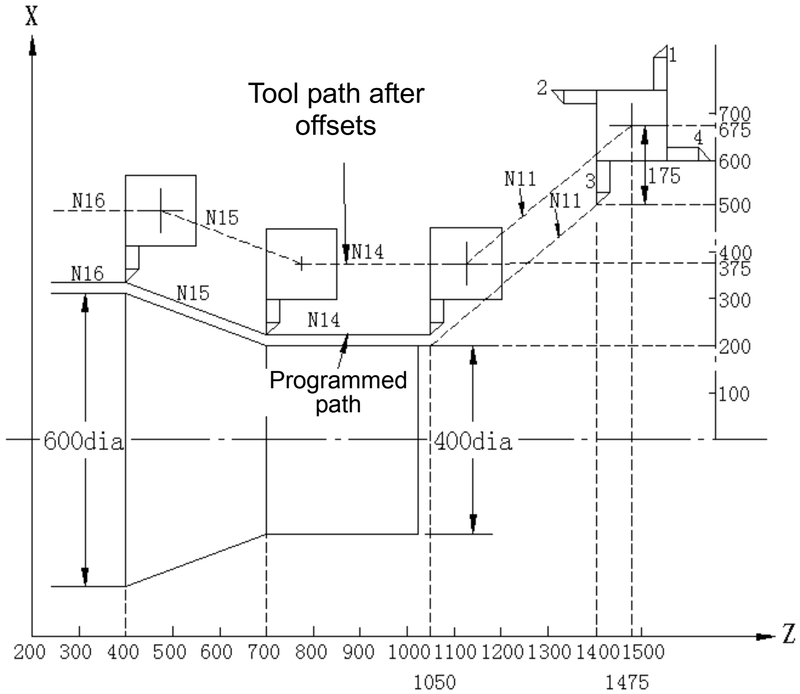
N222 G96 S100;　　　100m/min

⋮

N333 G97;　　　　　X rpm.

X is the spindle speed before N333 block. The spindle speed never changes when G96 turns into G97 mode. The S value of G96 is enabled when G96→G97. When the S is not specified, the S=0 m/min is applied (feed/min)

2.8.2.5 Example



N8 G00 X1000.0Z1400.0

N9 T33

N11 X400.0Z1050.0

N12 G50S300 (max. spindle speed)

N13 G96S200 (Surface speed 200m/min)

N14 G01 Z 700.0F1000

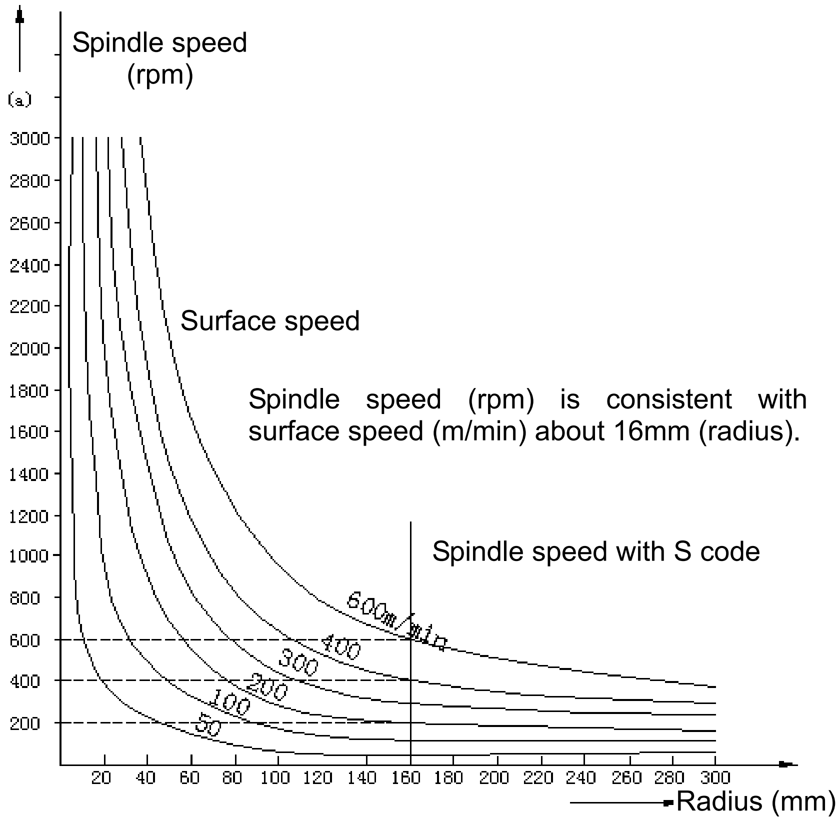
N15 X600.0Z 400.0

N16 Z...

The spindle speed calculated by NC system is proportional to the surface speed of coordinate value programmed position of X axis. When offset is valid, the surface speed is not calculated by the coordinate of X axis after offset. For the end of N15 on the above illustration, the speed is 200m/min. in 600 diameter(it is tool nose, not tool-post center)

If X axis coordinate value is negative, NC system calculates the surfaces speed by absolute value.

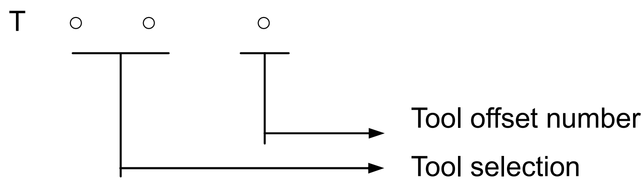
2.8.2.6 Relationships Between Spindle Speed and Surface Speed



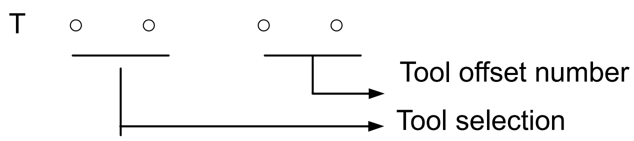
2.8.3 Tool Function

The digit followed the T code indicates the tool. A part of data is also used to indicate the tool offset number of offset amount. The following two descriptions are selected with parameter ＃8.3 (T2D).

(1) T2+1 setting



(2) T2+2 setting



Refer to the manual issued by the machine tool builder for corresponding T code and tool amount. Example (T2+2)

N1 G00 X1000 Z1400；

N2 T0313; …… (The 3rd tool and the 13th offset value are selected)

N3 X400 Z1050；

Tool selection number is propagated to the machine by BCD code (with strobe signal).

Note: Some machines is specified to a tool selection with a one-digital

2.8.4 Miscellaneous Function (M Function)

When a two-digit is specified with the following address M, a two-digit BCD code and a strobe signal are sent to machine tool side. These signals are used for controlling the ON/OFF of machine function. One M code is specified in one block. When two or more M codes are specified, only the last one is effective. Different M code selection depends on different machine, refer to the user manual issued by machine tool builder for details.

The following M code includes exceptional meaning.

2.8.4.1 M02，M30：End-of-program

(1) It is indicates that the main program is end.

(2) The cycle operation stops, and NC system resets. (refer to the manual issued by machine tool builder)

(3) When M30 is operated in Auto mode, it makes the program stop and the cursor return to the beginning of program.

2.8.4.2 M00：Program Stops

The cycle operation stops after M00 executes. When program stops, all the modal information and single program operation are invariable. They are started by a specified NC; the cycle operation is then restarted. (It depends on the machine builder).

2.8.4.3 M01：Stop Selection

Like M00, when the OPTIONAL STOP key on machine operator panel is pressed, the cycle operation stops after the block included M01 is performed. And, when OPTIONAL STOP is pressed once again, the cycle operation will be executed continually

2.8.4.4 M98：Subprogram Call

This code is used for calling subprogram, see the section 2.9.

2.8.4.5 M99：End of Subprogram

Subprogram end can be indicated with M99. Performing M99 and turn to the main program. See the section 2.9.

Note 1: The block followed M00, M01, M02 or M30 can not be read into the buffer.

Note 2: When M98 or M99 is performed, the code signal and selection signal are not propagated.

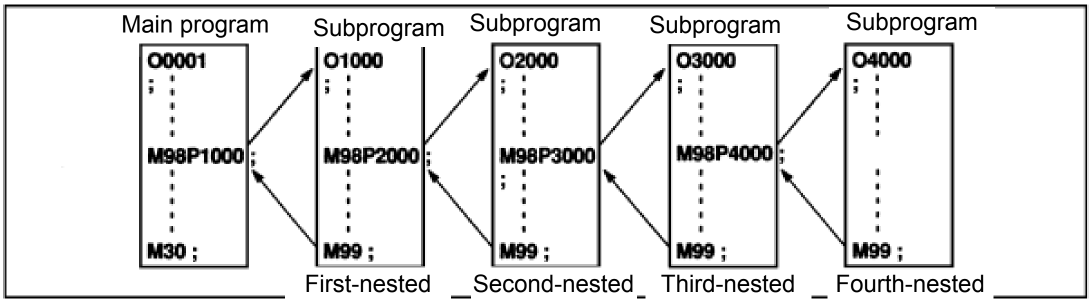
Note 3: The other M codes are treated with machine tool other than M99 and M98 codes. Because different machine tool builder has different configurations, please refer to the manual issued by the machine tool builder for details.

2.9 Subprogram

Some canned sequence is included or the modal displays repeatedly in a program, the sequence and modal can be regarded as subprogram to store to the CNC memory and simplify the program.

The subprogram can be called in memory working mode. A subprogram can call others subprogram.

When main program calls a subprogram, it is regarded as one step call. The occasion of performing of four-nested call subprogram is as follows:



A subprogram can be called repeatedly with a calling command, and a calling command can be repeatedly called for 9999 times.

2.9.1 Execution of Subprogram

O(:)X X X X;

………………………………;

………………………………;

………………………………;

M99;

The subprogram can be specified after “O” (EIA) or “:” (ISO) at its beginning. M99 might not to be specified with single block at the end of subprogram.

(For example)

X………………M99;

How to input the subprogram into the memory, see the operation section (4.4.15~4.4.17) for details.

Note: In order to make NC program is harmonized with the others systems. The subprogram of previous block also can be written into:

“N X X X X X “instead of O or (:),

The system records the digit followed with N is regarded as subprogram.

2.9.2 Performance of Subprogram

The subprogram is performed when it is called with main program or other subprogram.

The following specifications are employed when calling the subprogram:

M98P…………………L ……………;

Times for subprogram

Subprogram number

The subprogram is repeatedly once when “L” omits.

(For example) M98P 1002 L5;

The subprogram should be called for 5 times when the command no.1002 subprogram is performed.

Calling subprogram command (M98P_L_) and move the command can be indicated into the same block.

(For example) X1000 M98 P1200;

In this example, subprogram 1200 calling is performed after executing the movement of X axis.

The following sequence is from main program calling to subprogram calling:

Main program	Subprogram
N0010-----------------；	O1010---------------；
N0020-----------------；	N1020---------------；
N0030 M98 P1010 L2;	N1030---------------；
N0040-----------------；	N1040---------------；
N0050 M98 P1010；	N1050---------------；
N006------------------；	N1060------------M99；

When a subprogram calls another one, the execution process of subprogram is same as above figure.

Note 1: M98 and M99 signals are not propagated to machine tool.
Note 2: If the subprogram specified with P can not be found, the no.78 alarm occurs.
Note 3: Subprogram calls command "M98P_", it can not be input from MDI. Preparing the following main programs with MDI, see the following:

O XXXX；
M98 PXXXX；
N02；

And then, it is performed with Auto mode.

Note 4: Single block is disabled when it stops in the block of M98P_; M99; when M98, M99 block is included the address other than O, N, L and P, the single block is enabled when it stops.

2.9.3 Control Method of Subprogram

There are some especial usages, see the following:

a) When the sequence number is specified with address P at the last block of the subprogram, the subprogram does not return to the main program after ending to call the next block of subprogram. It returns to the block specified with the sequence number which is specified with address P, but, this function is only effective in Auto mode.

The return time of the specified block is usually longer than the normal time.

Main program	Subprogram
N0010--------------；	O1010--------------；
N0020--------------；	N1020--------------；
N0030--------------；	N1030---------------；
N0040 M98 P1010 ；	N1040--------------；
N0050-------------；	N1050--------------；
N0060-------------；	N1060--------------；
N0070-------------；	N1070 M99 P0070；

Note: If the return objective is an independent block of M98/M99 (The addresses are not command other than O, N, L and P); when it returns from a subprogram, it will be omitted the M98/M99 block to perform next block consecutively.

For example:

```
O0100                 O0200                     O0300
  ┆                     ┆                         ┆
  ┆                   N201 M98 P0300;             ┆
M98 P0200;            N202 M98 P0400;             ┆
                      N203……………;              M99 P0202;
                        ┆
                      M99;             Machining N203, not N202
```

So, the block of specified sequence number with M99PXXXX is same when it is a sequence number.

For example

```
  O0100;              O0200;                  O0300;
    ┆                   ┆                       ┆
    ┆                 N201 M98 P0300;           ┆
  M98 P0200;            N202;                 M99 P0200;
 N98 P0400;
 N203……;
                        ┆
 M99;
```

b) If a M99 command executes in main program, the control will return to the beginning of main program therefore.

For example, a “M99” is inserted in the fit position of main program; the block, which the optional block skip function is closed, the control returns to the beginning of main program when M99 executes.

If the optional block function switch opens, the “/M99;” is then omitted and control to the next block.

The control can not return to the beginning of the block if “/M99Pn;” is inserted, it returns to the block which the sequence is “n”

N0010-------------;

N0020-------------;

N0030-------------;

Close the skip optional block switch

N0040-------------;

N0050-------------;

N0060-------------;

/0070 M99P0030;

Switch on the skip optional block

N0080-------------;

N0090 M02 ;

c) Subprogram and main program are indexed from MDI (See the section of index of Chapter Four 4.4.14). So, a subprogram can be performed from beginning.

In this case, the M99 command is performed; the control will return to the beginning of subprogram and perform repeatedly.

In the above operation, if you want to stop the block at a fit position, you can insert the "/M02;" or "/M30;" in this fit position. When the skip optional block switch is power-on, the above command executes, if you want to end it, cutting the switch.

N1040---------------;

N1050---------------;

N1060---------------;

/N1070 M02;

N1080 M99P105

d) M99Lα;

When executing the above commands, the subprogram call repetition times L is compelling changed into α times. If the skip optional switch is closed in the following program, the repetition times will become into zero when the subprogram ending command (M99) is executed, the main program is then performed.

Main program Subprogram

M98P_L99;

/M99L0;

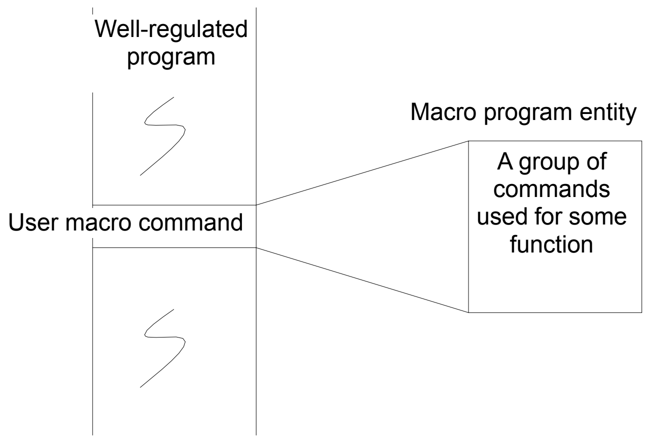
M99;

2.10 User Custom Macro Program

2.10.1 Brief

The function of user macro program A and B are basically same, and their differences are described in Section 10.10

Some kind of function is composited by some one group command, as like the subprogram restores into the memory. These memory functions are represented with a command. So, the function performed only needs to describe its representation command. This kind of commands registered are regarded as “Macro program body” and its representative commands are called “User macro commands”, the macro program body can be simplified as Macro program. And the user macro command can also be called Macro program command.

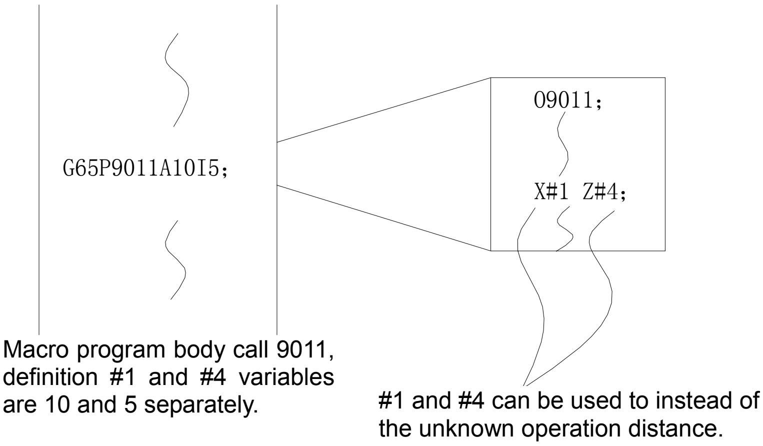


The programmer needs to remember the macro command only, it is not necessary to remember the command in the macro body.

There are three essentials for the user macro program: the variable can be applied into the macro program body, the user macro program can be finished the operation in terms of the variable, and the actual value can be evaluated the variable for the user macro command.

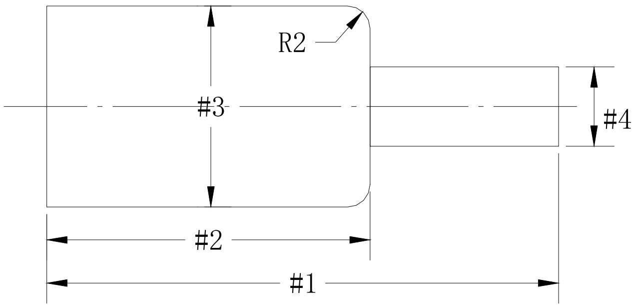
Note: Each machine tool builder should store their user macro program and save them into the memory.

Suppose that the PCB needs to change due to the malfunction. In order to rapidly check the malfunction, GSK service personnel or end user who can bear the maintenance and they should understand the content of user macro program.



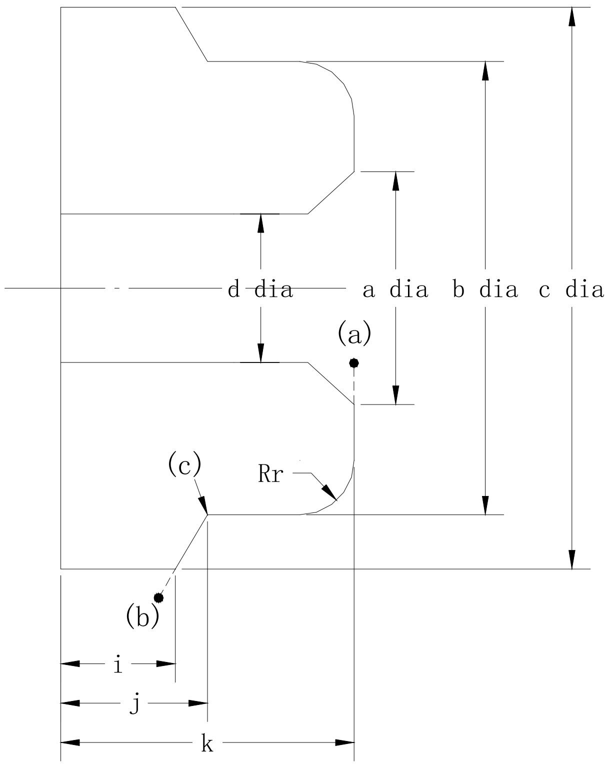
The current function can be formed when programming the macro program using certain function. For the changeable or unrecognized data which can be written into the program using variable. One of group technic can be offered.

Same workpiece can be collected into one group; a current user macro program can be programmed using the variable in a fit group. With the same method, the other workpiece in the same group is not needed to program. The programmer evaluates the actual value to the variable.



The workpiece, which the figure is same but the dimension is different, is not needed to program again any more when machining, as the above figure shows. If the variable used with # 1 to # 4 are performed the program to the macro program body. The actual value evaluates to the variable is allowed

Here, we will make a user macro program to finish the following workpiece machining from (a) to (b).



The following program can be read as:

Close to (a)   G00 X d Z k;

G01 X b R r;

Z j ;

Cutting from (a) to (b): c+3.0 chamfering is in the (b). The incremental value is W from (c) to (b) during Z= (c-b+3.0)* (i-j) / (c-b). However, X (c+3.0) Ww;

The macro program body uses variable #n. the #n is corresponding with the following letters.

A——————————#1
B——————————#2
C——————————#3

D——————————#7

I——————————#4

J——————————-#5

R——————————---#18

K——————————#6

W——————————#23

So, the actual user macro program body is as follows:

The macro program number is needed at the beginning; the M99 is needed to apply at last.

O9801; the user macro number is 9801

G00 X# 7 Z # 6;

G01 X # 2 R # 18;

Z # 5；

#23 = [#3 - #2 + 3.0]*[#4 - #5] / [#3 - #2]；

M99; the end of macro

A workpiece can be actually cut from (a) to (b), and the a, b and c should be evaluated. These values are specified to call the user macro program body during use the macro program command.

G65 P9801 B120.0 C150.0 D52.0
R10.0 I20.0 J30.0 K58.0;

A can be omitted in above description.

Normally, F, S and T functions and the coordinate system setting can be specified before specifying this command. The workpiece is same to the other types but the value different which should be machined to change the variable of macro program command.

2.10.2 Variable

The variable can instead of the value in macro program. After the variable is used, different values are evaluated with the variable when calling macro program. The macro program owns the ability of versatility hereby. Many variables can be employed simultaneously; each variable value can be specified with one variable.

2.10.2.1 Representation of Variable

The variable is indicated with # and its following digits. See the following below.

#i (i = 1，2，3，4……….)

For example 2. 10.2.1 #5

#109

#1005

It is also can be indicated with the following <Expression>

[< Expression>]

For example 2.10.2.2 # [#100]

[#1001-1]

[#6/2]

The variable #i can be replaced with variable # [<Expression>] in the following explanations.

2.10.2.2 Reference Variable

The value following the address can be replaced with variable.

<Address>#i or <Address>- # I it indicates that the variable or the complement of the variable is regarded as the command value of this address.

For example 2.10.2.3

When the F#33 #33 = 1.5, it is same to the F1.5.

When the Z-#18 #18 = 20.0, it is same to the Z-20.0 command.

When the G#130 #130 = 3.0, it is same to the G3.

(1) The variable is forbidden to use after address /, :, O, N.

For example: #27 and N#1 can not be employed. n (n=1~9) in the block skip/n can not be regarded as variable.

(2) Variable number can not be replaced with variable.

When the 5 in #5 is replaced with #30, it can not be written into # # 30, it is instead of # [#30].

(3) The variable value can not exceed the max. value specified with each address. When #140=120, the M#140 exceeds the maximum value (M code must be less than 99).

(4) When the variable is regarded as the data of address, the digits following with decimal point are rounded off.

(5) Using the following <Expression>. The subsequent digit can be indicated with <Expression>. Specifying <Address> [<Expression>] or <Address> - [<Expression>] means that the value of [<Expression>] or its complement is the digit of this address. Note that, the constant number without decimal point in [ ] is regarded as the number at the end with decimal point.

For example 2.10.2.4 X [#24+#18*COS (#1)]

Z－[#18 + #26]

2.10.2.3 Undefined Variable

The variable value undefined the variable is <Vacant>. Variable #0 is employed as its value is <Vacant>.

Characters of undefined variable see the following:

(1) Citation

The address is omitted when the undefined variable is quoted.

When #1=<Vacant>	When #1=0
G90X100Y#1	G90X100Y#1
↓	↓
G90X100	G90X100Y0

(2) Operator

The 0 value is same other than the value is evaluated with <Vacant>

When #1=<Vacant>	When #1=0
#2=#1 ↓ #2=<Vacant>	#2=#1 ↓ #2=0
#2=#1*5 ↓ #2=0	#2=#1*5 ↓ #2=0
#2=#1+#1 ↓ #2=0	#2=#1+#1 ↓ #2=0

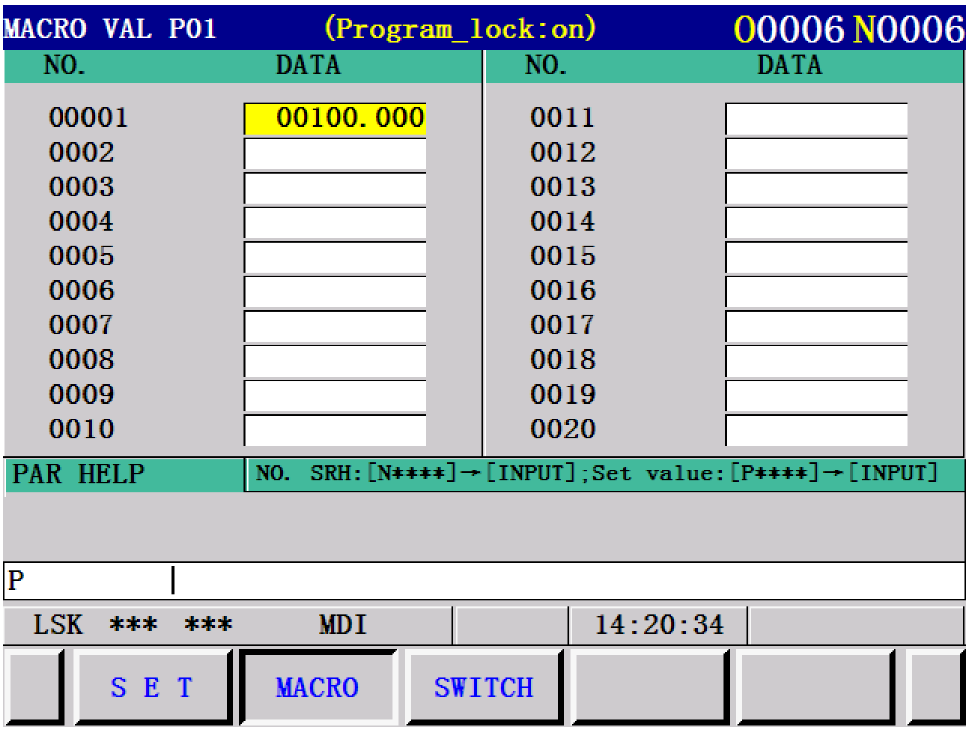
(3) Conditional expression

<Vacant> in QE and NE is different 0.

When #1=<Vacant>	When #1=0
#1 EQ #0 ↓ Established	#1 EQ #0 ↓ Unestablished
#1 NE 0 ↓ Established	#1 NE 0 ↓ Unestablished
#1 GE #0 ↓ Established	#1 GE #0 ↓ Unestablished
#1 GT 0 ↓ Unestablished	#1 GT 0 ↓ Unestablished

2.10.2.4 Display and Setting for Variable Value

Variables can be displayed on GSK983Ta's LCD and performed with MDI mode as follows:



About the detailed operation, refer to the user macro variable and setting in Section 4.4.8.2.

2.10.3 Kinds of Variable

In terms of variable number, the variable can be divided into: local, common and system variables. Usage and character of each variable is different.

2.10.3.1 Local Variable ＃1～#33

Local variable is that the local variable employed in program. That's it to say, the variable #i is used by calling in program in any case, and the local variable #i is used by calling in program in another place, both of them are actually different. So, if macro program A calls macro program B in multiple calls, the value of the local variable used in macro A will not be destroyed due to it is employed in macro B.

Local variable can be transferred and assigned by arguments, for the corresponding address of arguments, refer to Section 2.10.7, the initial status of the variable not be transferred is <Vacant>, the user can use it freely, the local variable ＃1～#33 will be cleared to <Vacant> when power OFF.

2.10.3.2 Common Variable ＃100～＃149，＃500～＃509

Local variable, is the variable employed in local in program, the common variable are employed with main program and each macro which the subprogram calls from main program. So, the common variable # i computed the result from some one macro program can be employed in other macro program.

The usage of common variable can not be applied in system, user can employ it freely.

Common variables from #100 to #149 are cleared into <Vacant> when the power is turned off, but from #500 to #509 will not be cleared.

2.10.3.3 System Variable

System variable has canned usage in system, it is used for reading and writing the data in NC, such as the tool offset and current position data. Also, it is the signal changed between machine programmable controller (PLC) and user macro program.

(1) Interface signal # 1000 to # 1015, # 1032, # 1100 to # 1115 and # 1132.

[Input signal]

The state of interface input status can be got by the value which is read from system variable # 1000 to # 1032.

System variable	Input interface signal
＃1000	2^0 UI0
＃1001	2^1 UI1
＃1002	2^2 UI2
＃1003	2^3 UI3
	2^4 UI4

# 1004	2^5 UI5
# 1005	2^6 UI6
# 1006	2^7 UI7
# 1007	2^8 UI8
# 1008	2^9 UI9
# 1009	2^{10} UI10
# 1010	2^{11} UI11
# 1011	2^{12} UI12
# 1012	2^{13} UI13
# 1013	2^{14} UI14
# 1014	2^{15} UI15
# 1015	

Variable value	Input signal
1	Connection close
0	Connection open

The read variable value 1.0 or 0.0 is regardless of unit system. So the unit system should be considered.

All the input signals can be read once by reading the system variable # 1032.

$$\#1032=\sum_{i=0}^{15} \#[1000+i]*2^i$$

System variable # 1000~# 1032 can not be employed on the left of operation command.

[Output signal]

Interface output signal can be sent out by evaluating to the system variable # 1100~ # 1132.

System variable	Output interface signal
# 1100	2^0 UO0
# 1101	2^1 UO1
# 1102	2^2 UO2
# 1103	2^3 UO3
# 1104	2^4 UO4
# 1105	2^5 UO5
# 1106	2^6 UO6
# 1107	2^7 UO7
# 1108	2^8 UO8
# 1109	2^9 UO9
# 1110	2^{10} UO10
# 1111	2^{11} UO11
# 1112	2^{12} UO12
# 1113	2^{13} UO13
# 1114	2^{14} UO14
# 1115	2^{15} UO15
Variable value	**Output signal**
1	joint close
0	joint open

Overall interface signals can be input once by evaluating to the system variable # 1132.

$$\#1132=\sum_{i=0}^{15} \#[1100+i]*2^i$$

The value of system variables # 1100 ~ # 1132 are sent out at last, which are saved as 1.0 or 0.0.

(Note) The values other than 1.0 or 0.0 are evaluated in # 1100~ #1115, see the following.

<Vacant> regards as 0

Other than <Vacant> regards as 1

It is indefinite when the value is less than 0.00000001.

(2) Tool form offset value from #2701 to # 2964, tool wearing offset value from # 2001 to # 2364.

Tool form offset value from # 2701 to # 2964 are read out, the workpiece coordinate system offset value can gain the offset value or deviation value using system variables of # 2501, # 2601. The offset value or deviation value can be modified by evaluating to the system variable #i.

X axis tool position offset value is radius value when it is specified with radius value, and it is diameter value when specifying with diameter value.

	Tool offset number	**Tool form offset amount**	**Tool wearing offset amount**
X	1 ⋮ 32	＃2701 ⋮ ＃2732	＃2001 ⋮ ＃2032
Z	1 ⋮ 32	＃2801 ⋮ ＃2832	＃2101 ⋮ ＃2132
R	1 ⋮ 32	＃2901 ⋮ ＃2932	＃2201 ⋮ ＃2232
T	1 ⋮ 32	＃2301 ⋮ ＃2332	＃2301 ⋮ ＃2332

Workpiece coordinate system offset	**Offset value**
X	#2501
Z	#2601

When only tool position offset is applied, the system variable using for tool position offset and the variable of tool form offset are same.

#2000, #2100, #2200 and #2300 are read only, and their values are always 0.

[Example 10.3.2] #30 = #2005

X axis tool wearing offset value with the offset number 5 is evaluated to the variable #30.

When the offset value is 1.500mm, #30 = 1.5, when the offset value is 0.1500inch, #30=0.15

＃2210 ＝ ＃8

Cutter compensation (R) with offset number 10 is modified into the same value with variable #8.

(3) Alarm #3000

When an error in macro program is checked, the device enters alarm status. Substituting an alarm number into the system variable #3000, the alarm LED displays after the previous block is processed, the device is then on the status of alarm.

#3000= n (Alarm information)

The alarm number which is never used before is regarded as the alarm number setting in the user macro program. (n<200)

The alarm information with 26 letters at maximum can be set between the dwelling control and recovery control.

(4) CLK #3001, #3002

The values of system parameter #3001 and #3002 are read out the CLK, the time displays therefore. Substituting the value into the system variable value to preset the time.

Type	System variable	Unit time	Power on	Count
CLK 1	＃3001	1 ms	Resetting to 0	Always effective
CLK 2	＃3002	1 h	Similar to the power off	When STL signal opens

The accuracy of CLK is 16ms. An overflow returns to 0 when the CLK 1 is 65536ms. The CLK 2 is consecutively added to the preset. The CLK cannot be executed corrected when the max. time exceeds 9544 hours.

Example 2.10.3.3 Timer

A calling of macro command

G65 P9101 T (Waiting time) ms

Macro program: the initial setting sequence is as follows:

```
O9101；
    #3001 = 0；
    WHILE [#3001 LE #20] DO1;      : Waiting the setting time
    END1；
    M99；
```

(5) The single block stops, waiting for the completion signal control of miscellaneous function.

The single block stop function can be controlled; when substituting the following value to the system value #3003, the next block is consecutively performed till the following block is not finish the signal (FIN) of miscellaneous function (S, T and M). The distributed signal (DEN) is not sent out till the finished signal is not applies. Note that this point can not be specified which the miscellaneous function is not waiting for the finished signal.

＃3003	Single block stop	Signal after M function finishes
0	Enable	Waiting
1	Disable	Waiting
2	Enable	No waiting
3	Disable	No waiting

[For example 10.3.4] Drilling cycle (Incremental programming)

A calling command of macro program

G65 P9081 L (Repeated times) R (Point R) W (Point Z)

Macro program body is as follows:

```
O9081;              Single block stops
#3003 = 1；
```

G00W #18; #18 equivalent to R

G01W #23; #23 equivalent to W

G00W-[ROUND[#18] + ROUND[#23]]；

#3003 = 0；

M99；

(6) Feedrate, speed override and exact dwell checking #3004 are disabled.

As for the following block, when the following values are substituted in system variable # 3004. Feedhold and feedrate are disabled and the exact dwell checking is not performed. The following status are formed when the feedhold button is controlled during executing the block which the feedhold is ineffective:

1) When the feedhold button keeps at the detention status, feedhold stops after the feedhold is effective at the first beginning of block.

2) Feedhold button holds and then releases, the feedhold light is power on but not stops, till to the end of the first block after the feedhold is enabled.

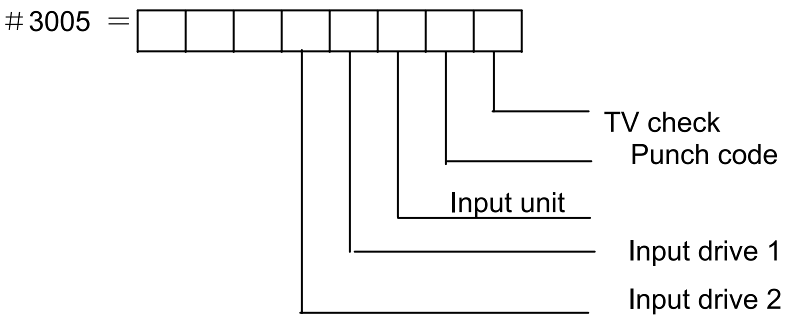
3)

＃3004	Feed hold	Feedrate	Accurate stop check
0	○	○	○
1	╳	○	○
2	○	╳	○
3	╳	╳	○
4	○	○	╳
5	╳	○	╳
6	○	╳	╳
7	╳	╳	╳

○：Function on ╳：Function off

(7) The variable #3005 of corresponding SETTING data.

The data in system variable #3005 are specified. SETTING data can be set. When the value of system variable #3005 is indicated with binary system, it is corresponding with the SETTING data.



[For example]

If #3005=5 executes, the setting data is as follows:

TV check=1

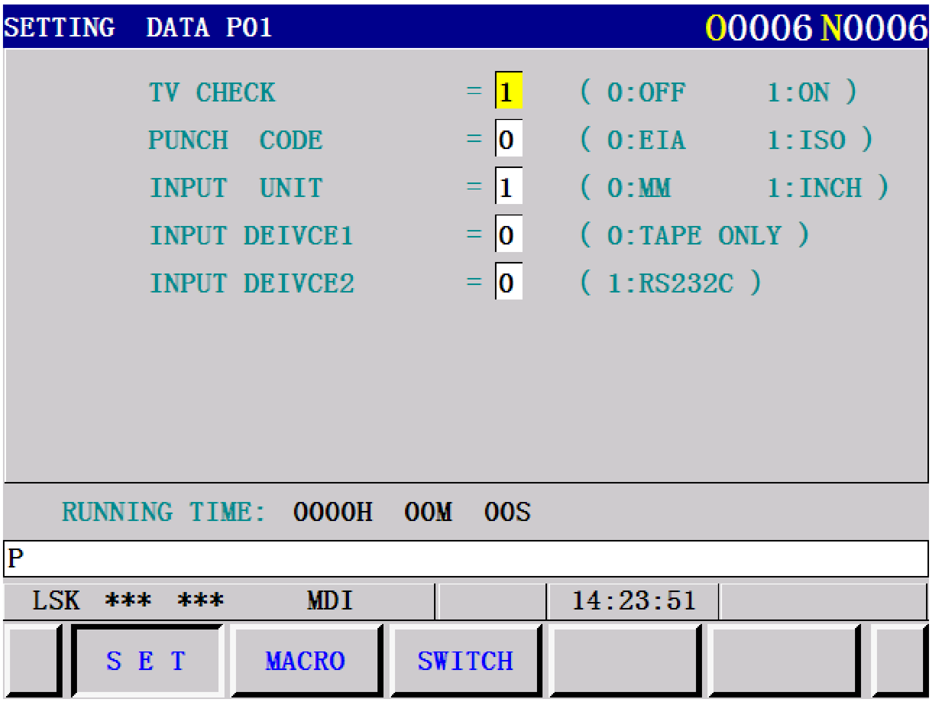
Punch code=0

Input unit=1

Input drive 1=0

Input drive 2=0

As follows:



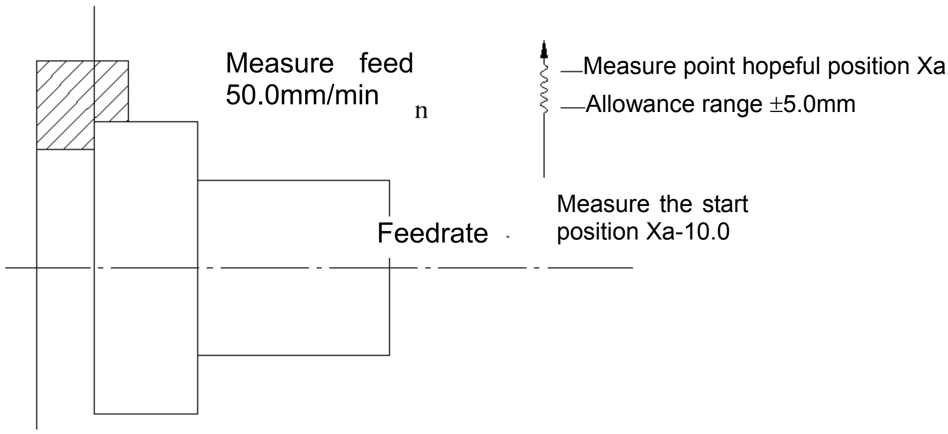
(8) Modal information #4001~#4120

The current value (the modal value commanded till to the previous block) of modal command can be computed by reading the system variables #4001~#4120 using the modal information.

The unit is effective when the unit is specified.

System variable	Modal information
＃4001	G code (01 group)
⋮	⋮
＃4021	G code (02 group)
＃4108	E code
＃4109	F code
＃4113	M code
＃4114	Sequence number
＃4115	Program number
＃4119	S code
＃4120	T code

[Example 2.10.3.5] X axis automatic tool offset



A calling command of macro program

G65 P9018 Xxa;

Macro program body is as follows:

O9018；	: Program number
#1 = #4001	: keep G code of group 01
G00 X[#24 – 10.0]；	: Measuring the start position at rapid traverse rate
G31 X[#24 + 5.0]；	: Measuring
IF [ABS[#5061 - #5081 - #24]LT5.0]GOTO 9018；	: Allowance range checking
#3000 = 80(PS)；	: PS alarm 80
N9018 #4 = BIN [BCD [#4120]AND255]；	: Tool compensation number
G10 P#4 X[5061 - #24]；	: Compensation
G #1 M99；	: Modal information recovery

System variable from #4001 to #4120 can not be used for the left of command operation.

(9) Position information #5001~#5102

Position information can be computed by reading out the value of system variable #5001~#5102. It is radius when the radius is applied; it is diameter when the diameter applies. The unit is mm or inches in terms of input system.

The system variable from #5001 to #5102 can not be used for the left of command operation.

System variable	Position information	Read in movement
＃5001 ＃5002	End coordinate in the section of X axis (A B S I O) End coordinate in the section of Z axis	Can
＃5021 ＃5022	Current coordinate in the section of X axis (A B S M T) Current coordinate in the section of Z axis	Cannot
＃5041 ＃5042	End coordinate in the section of X axis (A B S O T) End coordinate in the section of Z axis	Cannot
＃5061 ＃5062	X axis skip signal position (A B S K P) Z axis skip signal position	Can

＃5081 ＃5082	X axis tool offset value Z axis tool offset value	Cannot
＃5101 ＃5102	X axis servo position deviation Z axis servo position deviation	Cannot

Abbr.	ABSIO	ABSMT	ABSOT	ABSKP
Meaning	End point coordinate of top	Current coordinate command (Equivalent to POS. MACHINE)	Current coordinate command (Equivalent to POS.ABSOLUTE)	The effective position of SKIP signal in G31
Coordinate system	Work coordinate system	Machine coordinate system	Work coordinate system	System coordinate system
Tool offsets cutter compensation	Image tool nose position without considering	Considering tool nose base point position	Considering tool nose base point position	Considering tool nose base point position

Note: Tool position offset value is not the offset amount for the previous block but for the current block.

The position of SKIP signal is at the end of this block when the SKIP signal in G31 is not switched on.

For example the Section 2.10.3.6

The operation by some sequence is performed from the intermediate point of command moves to the some canned point of machine tool, and then it returns to the original position.

A calling command of macro program

G65 P9300 X (Intermediate point) Z (Intermediate point);

User macro program body specifying is as follows:

```
O9300;
    #1 = #5001;
    #2 = #5002;
    G00 X#24 Z#26;
    G04;
    U[xp - #5021] W[zp - ＃5022];
    ⋮     (Processing)
    ⋮
X#24 Z#26;
X#1 Z#2;
    M99;
```

2.10.4　Operation Command

Each operation can be performed during variable. The operation command programming is

same to the common calculation

#i = <Expression>

The <Expression> on the right of operation command is composed with constant, variable, function and operator. #i and #k can also be replaced by constant. The constant without decimal point in <Expression> is regarded as the last constant of decimal point.

2.10.4.1 Variable Definition and Replacement

＃i ＝ ＃j； Definition and replacement

2.10.4.2 Arithmetic Addition

＃i= #j + #k And

＃i = #j - #k Subtraction

＃i= #j OR # k Logic OR (Each one is “OR” together in 32 digits)

＃i = #jXOR #k Logic exclusive OR (Each one is “exclusive or” in 32 digits

2.10.4.3 Multiplication Calculation

＃i= #j × #k Product

＃i= #j ÷ #k Quotient

＃i= #j AND # k Logic OR (Each one is corresponding for 32 digits)

2.10.4.4 Function

＃i = SIN[#J] Sine (Unit degree)

＃i = COS[#J] Cosine (Unit degree)

＃i = TAN[#J] Tangent (Unit degree)

＃i = ATAN[#J]/ [#K] Arc tangent (Unit degree)

＃i = SQRT[#J] Square root

＃i = ABS[#J] Absolute

＃i = BIN[#J] Conversion from BCD to BIN

＃i =BCD[#J] Conversion from BIN to BCD

＃i = ROUND[#J] For integration (Round off)

＃i = FIX[#J] Values following the decimal point are casted out

＃i = #i=FUP[#J] Round off for the following of decimal point (Addition 1)

Note: Usage of Function ROUND

(1) The common decimal point is round off, when the operation command OR IF, WHILE are

used in conditional expression.

[For example] #1=ROUND [1.2345]

#1 becomes 1.0

IF[#1 LE ROUND [#2]] GOTO 10;

When #2 is 3.567, ROUND [#2] becomes 4.0.

(2) When address command is used, the least setting unit in this address is rounded off.

[For example]

G01 X [ROUND [#1]];

If #1 is 1.4567, the least setting unit of X is 0.001; this program becomes G01 X 1.457

But, in this example, it is same to G01X#1;

ROUND in the address command is mainly used in the following occasion:

[For example] The program returns to the beginning point only moving the incremental #1 and #2

```
N1 #1 = 1.2345;
N2 #2= 2.3456;
N3 G01 X#1 F100;          : X move to 1.2345
N4 X #2;                  : X moves to 2.3456
N5 X-[#11 + #2];          : # 1+#2 is 3.5801 X moves to -3.58
```

Because #1+#2=3.5801, so N5 can not return to the origin.

Suppose that the N5 is N5 X-[ROUND[#1]+ROUND[#2]];

It is equivalent to N5 X-[1.235=2.346]; the program can return to the initial point.

2.10.4.5 Composing of Calculation

The above operation and function can be combined. The priority sequence of calculation is function, multiplicative operation and additive operation.

Example) $\#1=\underbrace{\#2+\underbrace{\#3^{*}\underbrace{SIN[\#4]}_{\boxed{1}}}_{\boxed{2}}}_{\boxed{3}};$

$\boxed{1}$, $\boxed{2}$ and $\boxed{3}$ indicate operation sequence

2.10.4.6 Calculation Sequence Changes by Using [ ]

The prior part of operation sequence is included with [ ].

Bracket [ ], [ ] including function up to 5 layers at most

Example) #1=SIN [[[#2+#3]*#4+#5]*#6];

$$\#1=\mathrm{SIN}\ \underbrace{[\underbrace{[\underbrace{\underbrace{[\underbrace{\#2+\#3}_{\boxed{1}}]*\#4}_{\boxed{2}}+\#5}_{\boxed{3}}]*\#6}_{\boxed{4}}]}_{\boxed{5}};$$

$\boxed{1}$ to $\boxed{5}$ indicate operation sequence

2.10.4.7　Precision

The precision should be considered when programming and using the program of user macro program function

(1) Data format

The data used in user macro program is as the following floating-point format:

M*2E

Where: M: 1 digit symbol+31 digits binary number

E: 1 digit symbol+7 digits binary number

（2）Calculation precision

The following errors are generated after one calculation is performed. The error accumulates when computing repeatedly.

Calculation	Average error	Max. error	Error type
a=b*c	1.55×10^{-10}	4.66×10^{-10}	**Relative error (*1)** $\left\lvert\frac{\varepsilon}{a}\right\rvert$
a=b/c	4.66×10^{-10}	1.88×10^{-9}	
$a=\sqrt{b}$	1.24×10^{-9}	3.73×10^{-9}	
a=b+c a=b–c	2.33×10^{-10}	5.32×10^{-10}	**Min.** $\left\lvert\frac{\varepsilon}{b}\right\rvert$, $\left\lvert\frac{\varepsilon}{c}\right\rvert$ (*2)
a=SIN[b] a=COS[b]	5.0×10^{-9}	10×10^{-8}	**Absolute error (*3)** $\lvert\varepsilon\rvert$ **Degree**
a=ATAN[b]/[c](*4)	1.8×10^{-6}	3.6×10^{-6}	

(Note) Function TAN is realized with SIN/COS

2.10.4.8　Processing for Precision Decreasing

（1）Additive and subtractive

Note that the relative error cannot be guaranteed within 10^{-8} in additive and subtractive operation, when the absolute value applies to the subtractive operation.

For example, the actual values of #1 and #2 are as follows:

＃1=9876543210123.456

＃2=9876543277777.777

＃2-#1=67654.321

＃2-#1=67654.321 can not be gained when calculating the #2-#1.

Because the user macro program precision is 8 digits of decimal system, the values of #1 and #2 have the following lower precision:

#1=9876543200000.000

#2=9876543300000.000, (Because they are binary system number, it is less differences to the above values).

So, the bigger error will generated when ＃2-＃1= 100000.000.

(2) Logic operation

The error because of the EQ, NE, GT, LT, GE and LE are basically the same as the additive/subtractive.

Whether the #1 and #2 are equivalent is judged in above example.

IF [#1EQ#2] indicates that the judge may incorrect.

When error is calculated as follow, IF [ABS [#1-#2] LT5000]. If the difference of #1 and #2 is in the range of this error is regarded as equivalent.

（3）Floor function

Attention should be paid when using the down round numbers command.,

Example:

When #1=0.002 in #2=#2*1000 is calculated, result of variable #2 is not exact 2, may be 1.99999997.

When #3=FIX [#2] is specified, result of variable #3 is 1.0 rather than 2. Herein, correct the error firstly, and then perform the down round numbers or round-up to get the right result.

#3=FIX [#2+0.001]

#3=ROUND [2]

(4) Triangle function

The absolute error occurs in the triangle function, but the relative error cannot be less than the 10^{-8}, so, note that when the multiple-divided calculation performs and after the triangle function is calculated.

2.10.5 Control Command

The program schedule can be controlled using the following command.

2.10.5.1 Conditional expression

IF [<Conditional expression>] GOTO n

The block, its sequence number n begins perform from a same block, when <Conditional expression> is established, the n can be replaced with variable or [<Expression>], the next block is consecutively performed when the condition is not established.

If the IF [<Conditional expression>] is omitted, it is unconditionally branched to the block with

sequence number n.

<Conditional expression> are divided into the following items:

#j　EQ #k　　=

#j　NE #k　　≠

#j　GT #k　　>

#j　LT #k　　<

#j　GE #k　　≥

#j　LE #k　　≤

The <Expression> can be replaced with #j and #k. And the usable variable or [<Expression>] replaces n.

Note: The block with sequence number n will be performed after the GOTO　n, the sequence number should be placed at the beginning of the block.

The performance time of reverse direction difference is longer than the positive direction.

2.10.5.2　Cycle

When the specified conditions is satisfied, between DO and END blocks of WHILE are executed, otherwise, the system executes blocks following END, the command format is applied to the number following DO and END of IF statement, which is the label used to specify the program execution range. The label value is 1, 2, 3. P/S No. 126 alarm occurs when other labels except for 1, 2, 3 are used.

WHILE [<Conditional expression>] DO m (m=1, 2, 3)

┆（cycle block）

END m

The block is repeatedly performed from the next block of Dom to ENDm during the <Conditional expression> establishing, that is, judging the <Conditional expression> in Dom program. It performs from next block when <Conditional expression> is established; the next block of ENDm is performed when it is not established.

WHILE [<Conditional expression>] can also be omitted, like IF. The WHILE [<Conditional expression>] is repeated unlimitedly when it is omitted.

WHILE [<Conditional expression>] Dom and ENDm should be employed simultaneously. The mutual corresponding relationship is distinguished from the number m.

Example　Section 2.10.5.1

＃120 ＝ 1；

N1 WHILE [#120 LE 10] DO 1；

Chapter 2　Programming

Repeat for 10 times
N2 WHILE [#30 EQ 1] DO2

N3 END2；

Repeat during variable #30=1

#120 = #120 + 1；

N4END1；

Notice: The following precautions should be noticed when programming repeatedly;

(1) DOm should be specified firstly, the ENDm is then specified.

⋮

⋮

END1;

⋮ (Cannot)

⋮

DO 1

(2) DOm and ENDm should correspond one by one in a same program.

⋮

DO1； (Cannot)

⋮

DO1；

⋮

END1；

⋮

DO1；

⋮

⋮

END1； (Cannot)

⋮

⋮

END1；

(3) Same identification number can be used for many times.

⋮

DO 1；

⋮

END1； (Can)

⋮

DO 1；

⋮

END1；

⋮

(4) There are 3 layers for DO

⋮

DO 1；

⋮

DO 2；

⋮

DO 3；   (Can)

⋮

END3；

⋮

END2；

⋮

END1；

(5) The range of DO can not be crossed

⋮

DO 1；

⋮

DO 2；   (Can)

⋮

END1；

⋮

END2；

⋮

(6) The difference can be applied from the range of DO to the external.

⋮

DO1；

⋮   (Can)

GOTO 90000；

⋮

END1；

⋮

N9000……；

⋮

(7) The difference cannot be applied from the range of DO to the external.

⋮

GOTO 9000；

⋮

```
                DO 1；              (Cannot)
                ⋮
N9000……；
                ⋮
END1；
                ⋮
DO1；
                ⋮
                N9000……；         (Cannot)
                ⋮
END1；
                ⋮
GOTO 9000；
                ⋮
```

(8) User macro program body and subprogram can be called in the body DO. Layers of DO can be nested up to 3 inside the user macro program body and subprogram.

```
                ⋮
DO1；
                ⋮
                G65；               (Can)
                ⋮
                G66；               (Can)
                ⋮
                G67；               (Can)
                ⋮
END1；
                ⋮
DO1；
                ⋮
                M98；               (Can)
                ⋮
END1；
                ⋮
```

2.10.6 Programming and Storage of User Macro Program Body

2.10.6.1 Programming of User Macro Program Body

The format of user macro program body is same to the subprogram, the creation is as follows:

```
O (Program number);
Command
M99；
```

The program numbers are divided into the following parts:

(1) O1～O79999 Using for the program which can be stored, deleted and edited freely

(2) O8000～O8999 The program can not be stored, deleted and edited if the setting is not performed.

(3) O9000～O9019 Special macro program for the calling format

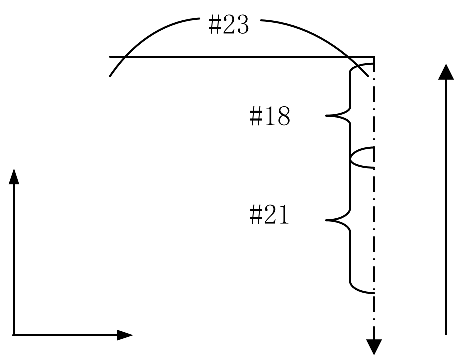
(4) O9020～O9899 The program can not be stored, deleted and edited if the setting is not applied.

(5) O9900～O9999 ROBOT operation program

Imaginary argument variable (It is used for accepting the data conversion of macro program calling command) is fixed. That is to say, the address of instruction parameter in macro program calling command should correspond one by one to variables in the user macro body (acceptor).

Example: Section 2.10.6.1

```
O9081;
G00W#23;
U#18;
G01U#21;
G00U-[#18 + #21];
M99;
```



2.10.6.2 Storage of User Macro Program Body

User macro program body is a kind of subprogram; the storage and operation are same to the subprogram.

The storage space for storing the macro program includes into the storage capacity of CNC.

2.10.6.3 Macro Statement and NC Statement

The following blocks are referred to as macro statements

(i) Blocks containing an operation command (such as logic operation =)

(ii) Blocks containing a control command (Such as GOTO, DO or END)

(iii) Blocks containing a macro call command (Such as G65, G66, and G67. The block of macro call of G code).

The program of non-macro program statement is called NC statement.

The macro statements have differences from NC statements, see the following.

(1) Usually, the single block not stops in single block mode.

(2) It is not regarded as [not move the block] in cutter compensation C.

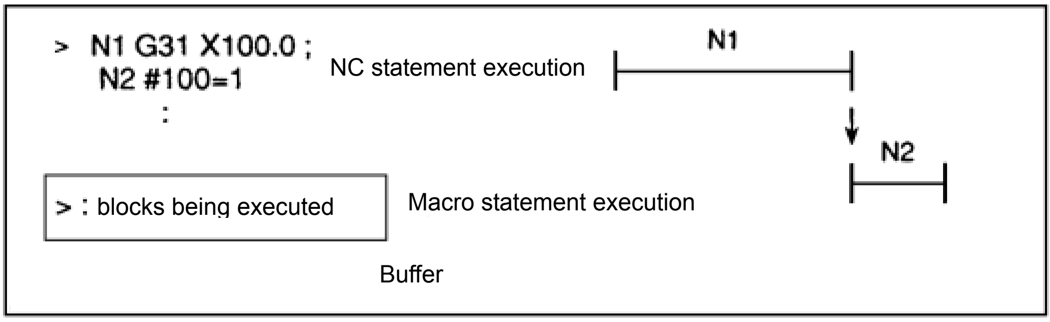
(3) Distinguishing from performing the time:

2.10.6.4 Macro Program Statement Execution

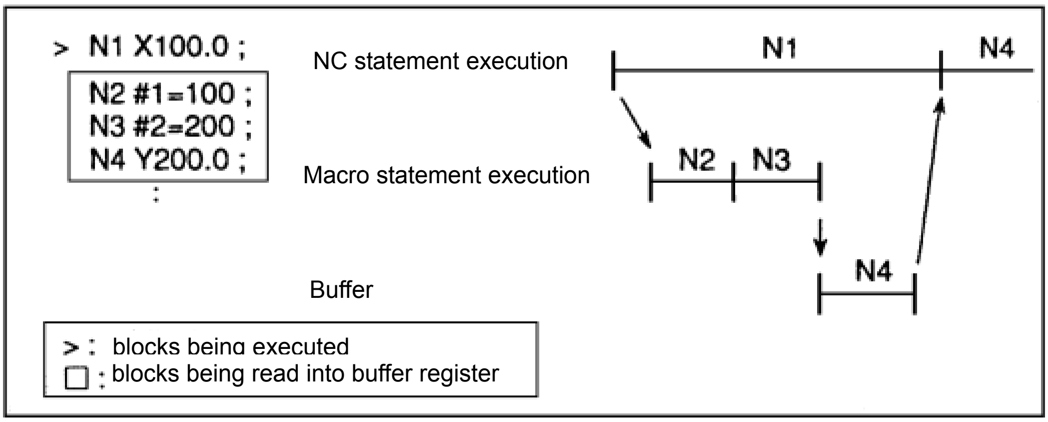
To smoothly machine, the CNC pre-reads the next NC statement to be executed, which is called buffer. In G41, G42, the NC pre-reads statements after NC of 2 or 3 blocks in advance, macro program statement of arithmetic expression and conditional jump are executed after they are buffered into register, besides blocks of M00, M01, M02 or M30 and other prohibited M codes, and the blocks including G31 do not reread.

Explanation:

① When the next block buffers (M code, G31 do not buffer)

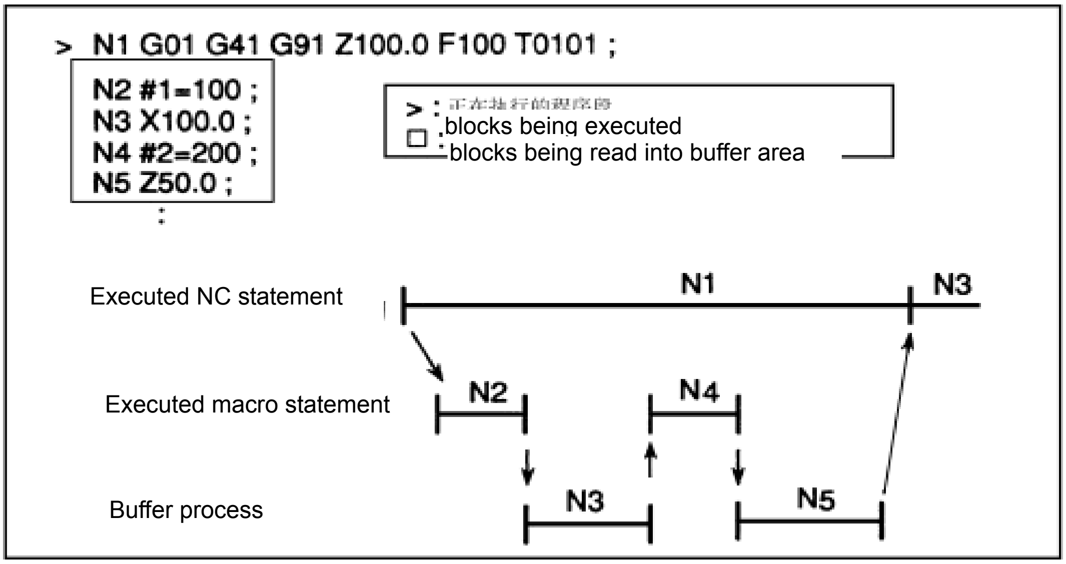


② Buffer the next block (normally reread the next block) in other modes besides tool radius compensation mode



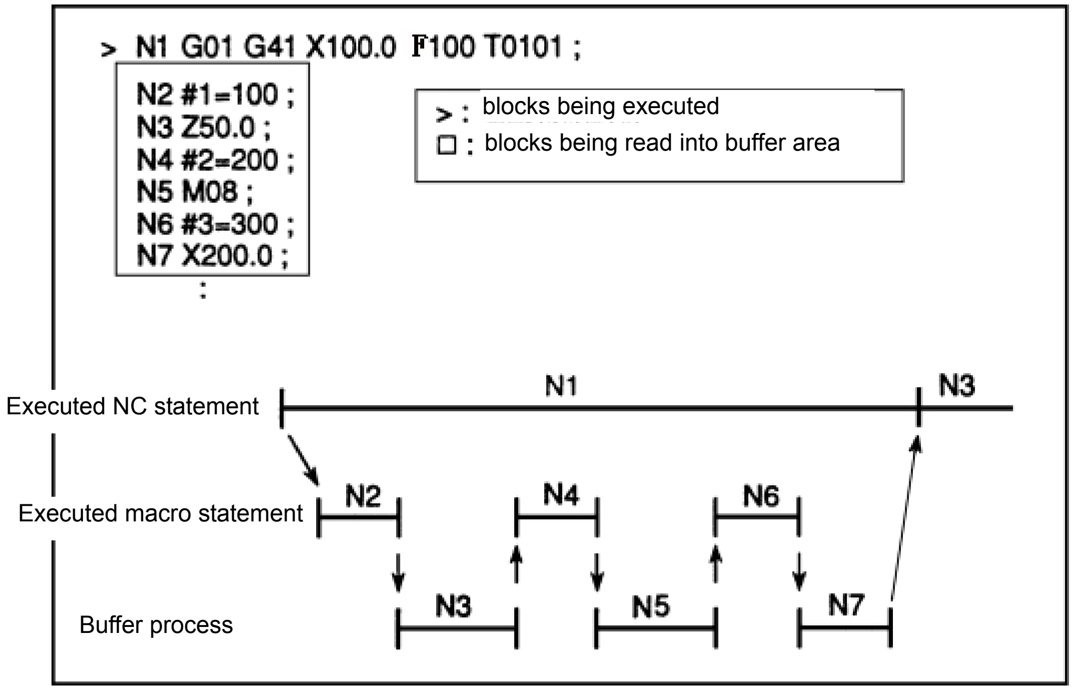
When N1 is executed, the next NC statement (N4) is buffered, macro statements between N1 and N4 are proceeded when N1 is executed.

③ The next block is buffered in tool nose radius compensation



When N1 is executed, NC statement of the next 2 blocks are reread into the buffer area, macro statements between N1 and N5 are proceeded when N1 is executed.

④ When there is no movement command in the next block in tool nose radius compensation (G41,G42)



When N1 block is executed, NC statement of the next 2 blocks are reread into buffer area. Because N5 statement has no movement command, its intersection cannot be counted, at the moment, NC statements of the following blocks (to N7) are read, macro statements (N2, N4, N6) between N1 and N7 are proceeded when N1 is executed.

2.10.7 Macro Program Call Command

The macro program can be called by the following method:.

- Macro program call
  - Non-modal call(G65)
  - Modal call(G66, G67)
  - Call macro program by G code
  - Call macro program by M code
  - Call subprogram by M code
  - Call subprogram by T code

2.10.7.1 Simply Call (G65)

The user macro program body is specified with P by executing the following command calls:

G65P (program number) L (Repeated time) <argument assignment>; when the macro program body must be transferred with arguments, the arguments were specified with <argument assignment>. There are two kinds of <argument assignment>. The arguments are the actual value for the variable in user macro program body.

Note: G65 must be specified before the arguments of its block, it is regardless of address.

<Argument evaluation> can be employed the negative/decimal point.

(1) Argument assignment I

A________B_________C_________D ……Z_________

All letters can be specified other than G, L, N, O and P. It is specified in terms of address form, and it is not necessary for the especial address sequence.

But, I, J and K must be specified by alphabetically.

B_____A_____D_____ …… I_____K_____…… Can

B_____A_____D_____ …… J_____I_____…… Cannot

The address evaluated in argument assignment **I** and the variable numbers used in macro program body have following corresponding relationships:

Argument assignment I address	Variable of user macro program body
A	#1
B	#2
C	#3
D	#7
E	#8
F	#9
H	#11
I	#4
J	#5

K	#6
M	#13
Q	#17
R	#18
S	#19
T	#20
U	#21
V	#22
W	#23
X	#24
Y	#25
Z	#26

(2) Argument assignment **II**

A____B____C____I____J____K____I____J____K____……

The arguments are specified other than in the address of A, B and C. Address I, J and K as a group can be specified the arguments for 10 groups. When multiple arguments are specified at the same address, they should have a line from the presetting sequence.

The address can be omitted if it is not necessary.

The address commanded in argument assignment **II** and the variable number of the variable used in macro program have following corresponding relationships:

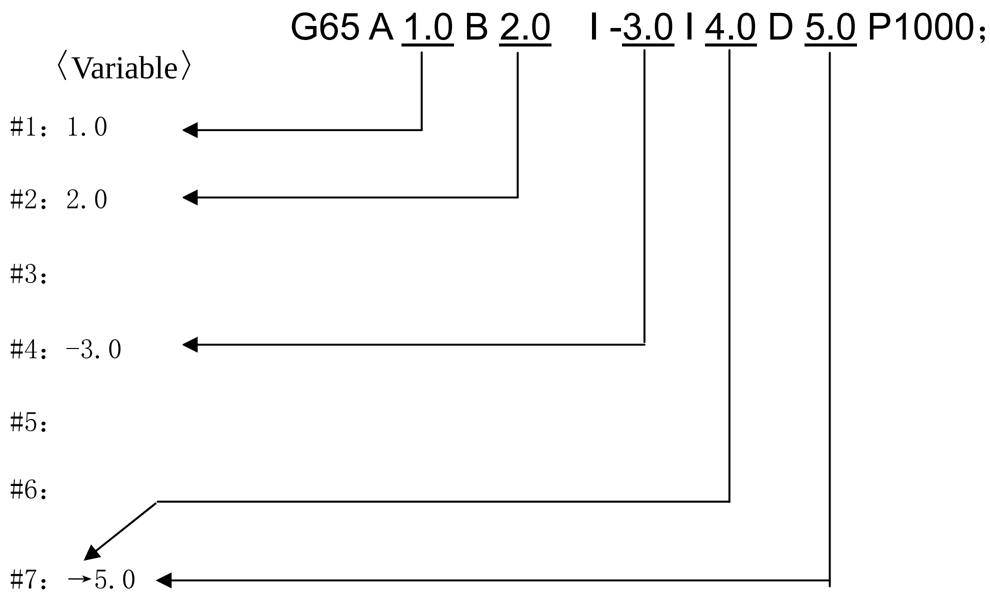
The subscript 1~10 of I, J and K are indicate the sequence of the commanded group.

Note: G65 must be specified before any arguments. (Argument assignment) can use negative sign, decimal point.

(3) Argument assignment I and II can be mixed use.

The specifying is not alarm even if arguments I and **II** are specified in G65 block simultaneously. As for the same variable, if the arguments I and **II** are specified at the same, the command specified at the back is effective.

For example



When the arguments I4.0 and D5.0 are specified by variable corresponding to #7 in above example, the latter D5.0 is enabled.

2.10.7.2 Modal Call（G66）

Macro call mode can be specified by performing the following commands. In macro call mode, calling for once the specified Marco program when performing the movable command for each time. G67 cancels modal call.

G66P (Program number) L (Repeated time) <Argument assignment>;

After G66, P specifies the program number of user macro program.

L specifies the repeated times from 1 to 9999. When L is omitted, L is considered to 1.

Argument assignment is same to the simple call.

#6：

Macro call mode is cancelled by performing the following command.

G67；

Note: G66 must be commanded before all the arguments in G66 block. Using Minus and decimal point in <argument assignment> is regardless of the address.

2.10.7.3 Multiple Call

Macro program can be called from macro program is like subprogram.

Call time, simple call and modal call can be realized up to 4 layers.

2.10.7.4 Multiple Modal Call

The specified macro program is called at once when the movement command is performed once in macro program modal call. When multiple modal macros are specified, the next macro program is

called for performing each movement command. The macro program call is effective after it is specified.

For example the Section 2.10.7.5

G66P9100；

Z10000； （1-1）

G66P9200；

Z15000； （1-2）

G67； ：P9200 Cancel

G67； ：P9100 Cancel

Z-25000； （1-3）

O9100；

X5000；

M99； （2-1）

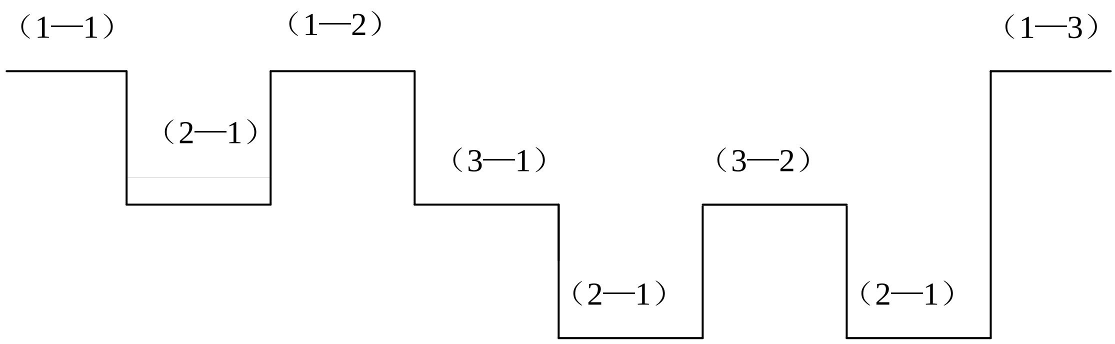
O9200；

Z6000； （3-1）

Z7000； （3-2）

M99；

Performance sequence (Block omittance without moveable)

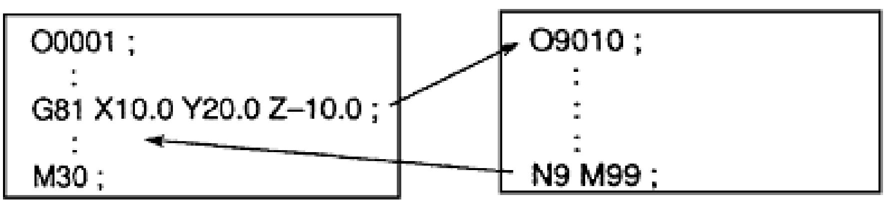


Note: The modal macro program can not be called depart from the macro program call mode after (1-3).

2.10.7.5 Macro Program Call with G Code

G code setting with parameter can call the macro program. That is: N_____G65P△△△△

<Argument assignment> can be replaced with the following simple command, and they perform the same operations



NC parameter 323＝81

The corresponding relationships of the G code use for calling and macro program number △△△△ called are set in parameter.

There are 10 in G01~G255 can use for calling the macro program other than G00.

These G codes can not be specified with MDI, like G65.

These G codes can not be specified in the called macro program, and can not be specified in the subprogram called with M code, T code either.

Setting the following parameters:

0	3	2	3		Macro program call: G code of 9010

0	3	2	4		Macro program call: G code of 9011

⋮ ⋮

0	3	3	2		Macro program call: G code of 9019

2.10.7.6 Subprogram Call with M Code

M code set with parameter can be called subprogram. That is: N_____G_____X Y________ ____ ……M98 P△△△△; the command can be replaced with the following simple one, but the operation is same.

N___G___X___Y___……M××；

Similarly to M98, subprogram displays at the COMND page, but MF and M codes are not delivered.

The corresponding relationships of M code XX called with subprogram and the program number △△△△ called by subprogram must be set into the parameter.

Up to three of M03~M97 can be employed for this kind of macro program call other than the M30, the MBUF1, MBUF2 of the parameter number 35, 36.

This command can be specified with MDI, but it can not be employed the arguments.

Subprogram call will not be performed when these M codes are specified into the macro program called with G code or the subprogram called with M, T codes. And these M codes are regarded as common.

Setting the following parameters

0	3	2	0		Macro program call: M code of 9001

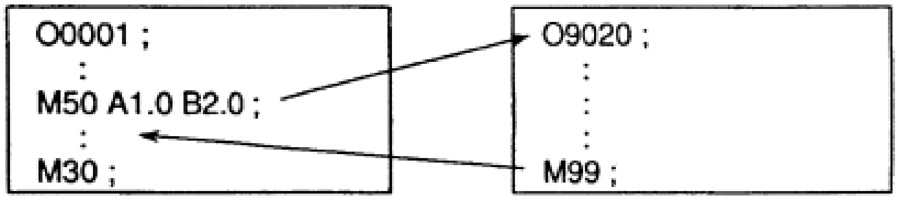
0	3	2	1		Macro program call: M code of 9002

0	3	2	2		Macro program call: M code of 9003

2.10.7.7　Macro Program Call with M Code

M code set with parameter can called macro program, that is N___G65P△△△△ <Arguments specifying>;

Same operations can be realized using the following commands:



NC parameter 43＝50

The corresponding relationships of the M code use for calling and macro program number △△△△ called are set in parameter.

Up to ten of M06~M255 can be called with macro program, except a part of specified M codes.

These M codes can not be specified with MDI, like G65, and they can not be specified in the macro program of G, M and T codes, can not be specified in the subprogram called with M or T code either.

Setting the following parameters:

	0	4	3		User macro program body call: M code of 9020

	0	5	2		User macro program body call: M code of 9029

2.10.7.8　Subprogram Call with T Code

T code calling subprogram command can be employed by setting parameter: N___G___X Z___ …… Tt;

It is same operations as the following two blocks:

#149=t；

N___G___X___Z___ …… M98　 P9000；

T code t is regarded as argument storing into the common variable #149.

T code displays on the page of COMND, but the TF, T codes are not output.

<u>This command can be specified with MDI, but it can not be specified in the same block which is called with M code.</u>

When the T code is specified, the subprogram call is not performed but same to the common T code in the macro program call with G code or subprogram call with M, T code.

Setting the following parameters:

0	3	0	6							TM CR	

Note: When TLCC of parameter number 8 is 1, the compensation is enabled in the block of T.

2.10.7.9 Decimal Point Position of Arguments

Generally, arguments are applied with decimal point, if the decimal point is not performed, the position of decimal point is regarded as the following:

Address	mm input	Inch input
A, D	0	0
B, C, H	3（2）	3
E	4（5）	6
F (In the mode of G99)	2（3）	4
F (In the mode of G98)	0（1）	2
I, J, K	3（2）	4
M, S, T	0	0
Q	0	0
R	3（2）	4
U, V, W	3（2）	4
X, Y, Z	3（2）	4

The values in the table are indicated the decimal point position is starting from the least effective digit.

As for the address E, F in the bracket ( ) is the parameter FMIC=1, it is the digit of following of decimal point when the other addresses are parameter MIC=1.

2.10.7.10 Distinguish from M98 (Subprogram call) and G65 (User macro program call)

(1) G65 can specify the arguments, but M98 can not.

(2) The command is performed other than M, P and L and convert to the subprogram in M98 block, but G65 is converted only. (The others are not performed)

(3) The single block stops when the address is not included O, N, P and L in M98 block. But G65 not stops.

(4) G65 changes the grades of local variable, but M98 not changes, that is: #1 before command G65 is different to the #1 inside the called user macro program body. And #1 before command G98 is different to the #1 inside the called subprogram.

(5) Up to 4 layers of call time is gathered with G65, but the G98 call time is 4 layers (When the user macro program A or B selection is applied).

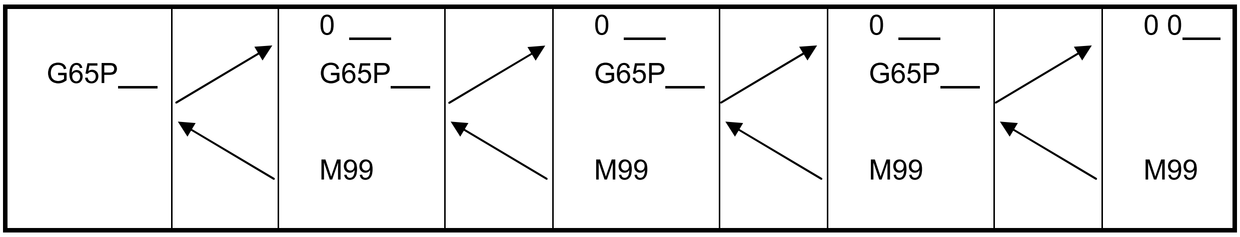
(6) When MDI operation is inserted in Auto, time of M98 is 4 layers in Auto, but it is separately reached to 4 layers in MDI mode, G65 arrives 4 layers in all modes.

2.10.7.11 Nestification and Local Variable of User Macro Program

Macro program is called once with G65, G66 or G code, the macro layer (level) is added 1, the level of local variable is added 1 simultaneously.

Namely: Macro program call and local variable have the following relationships:

Main program



Local variable

(Level 0)		(Level 1)		(Level 2)		(Level 3)		(Level 4)
#1		#1		#1		#1		#1
≀		≀		≀		≀		≀
#33		#33		#33		#33		#33

(1) Local variables from #1~#33 can be employed in main program (Level 0)

(2) The local variable (Level 0) is saved in main program as long as the macro program (Level 1) is called with G65; the local variables from #1~#33 are employed in new macro program (Level 1), and arguments can be entered from them [(3) is same].

(3) The local variables (Level 1, 2 and 3) are all stored when the macro program (Level 2, 3 and 4) is called for each time, and the new local variables are employed (Level 2, 3 and 4).

(4) The local variable stored in (2) and (3) is restored to the initial status as long as it returns with M 99 from each macro program.

2.10.8 Relationships with Other Functions

(1) MDI operation

Macro call, operation and control commands are not be commanded in MDI mode.

MDI can be inserted in macro program performing even if when the single block stops. (The command related to the macro program is exceptional)

If the MDI mode is operated, the macro program is not called even if it is in the macro program call mode.

(2) Sequence number index

The sequence number can not be indexed in macro program body.

(3) Single block

The blocks other than macro program call, operation command and control command can be stopped even if in macro program

The block of macro program call (G65, G66 and G67), operation and control commands are not stopped even if in the mode of single block.

But, after the following data or parameters are set, the single blocks are all stopped other than macro program call.

Using for the test of macro program body:

0	3	1	8			MCS						

						9						

0	3	1	9			MCS8						MCS7

Macro program statement in 01~7999, 09900~9999 are performed single block stop, when MCS7=1

Macro program statement in 08000～8999 are performed single block stop when MCS8=1.

Macro program statement in 09000～9089 are performed single block stop when MCS9=1.

When macro program single block stop is employed, it is regarded as unmovable in cutter compensation R, when the incorrect compensation may occur. (Strictly speaking, although the movement is specified and it is same to the movement value 0).

This setting is prior to the #3003 single block stop control, that is: if the MCS7, 8 and 9=1, even if the #3003=1 (or 3) single block stops and not to be controlled in the corresponding range of program number, MCS7, 8 and 9 are the parameter used with macro program checking, so, it is very necessary to set these parameters to 0 if the macro program checking ends.

(4) Skip optional block

When the / code occurs in the middle of <Expression> (On the right of arithmetic expression, or in the middle of [ ], it is regarded as division sign instead of skip optional block.

(5) Operation in the mode of edit

In order to not damage the user macro program body and subprogram stored due to the incorrectly operation, the following setting is as shown.

0	3	1	8		PRG 9							

0	3	1	9		PRG 8							

PRG8=1, User program body or subprogram of program number 8000～8999

PRG9=1, User program body or subprogram of program number 9000～9899 can not stored, deleted and edited, but, all the programs are deleted when the power is turned on, and the above limitation is not applied when single program punching issues.

(6) Program page displays depart from the edit mode.

Normally, when macro body and subprogram are called, the called program displays, the previous program can be kept performing the following settings.

0	3	1	8				MPD 9					

0	3	1	9				MPD 8					

When MPD8=1, MPD9=1, user macro program body and subprogram of the corresponding

program number 8000～8999 and 9000～9899 can not be displayed on the program screen external the edit mode.

（7） Reset

The clear status occurs using the reset, all of the local variable and common variable #100～#149 are eliminated into <Vacant>, but the system variable is not eliminated.

The resetting occurs depart from the MDI mode, the user macro program body, subprogram call status and DO status are all eliminated, and return to the main program, the call with MDI are eliminated when setting in MDI mode.

(8) Macro program statement and NC statement

The following blocks are regarded as macro program statement

a) Operation command (the block containing #)
b) Control command (The GOTO, DO or END block contains).
c) Macro program call command (The block of G code contains G65, G66, G67 or macro program call)

The block other than the macro program statement is regarded as NC statement.

(9) MDI intervention in automatic operation

When macro program call is applied with MDI intervention in automatic operation, the nesting number of macro program call and DO are called up to 4 layers when automatic operation starts consecutively at the same, DO calls 3 layers, the subprogram nesting number can be reached 4 layers in MDI mode.

(10) Composite canned cycle

Macro program statement (Control command, Operational command and Macro program call command) can not be employed in composited canned cycle (G71, G72 and G73)

(11) Program restarts page display

The MT code used is not displayed, as M98, when M code/T code calls to subprogram.

(12) Feed hold

Macro program statement can be stopped to perform using feed hold (It stops when resetting or the alarm occurs).

2.10.9 Especial Codes and Words in Macro Program

The following codes are applied other than the codes are applied in usual macro program.

（1）ISO

Meaning	8 7 6 5 4 3 2 1	Symbol
[	○ ○ ○ ○ ○ ○ ○	[
]	○ ○ ○ ○ ○ ○ ○	]
#	○ ○ ○ ○ ○	#
*	○ ○ ○ ○ ○	*

=	○ ○ ○ ○ ○○ ○	=
0	○ ○ ○ ○○ ○ ○	0
+	○ ○ ○ ○ ○	+

（2）EIA

Meaning	8 7 6 5 4 3 2 1	Symbol
[	○ ○ ○○	
]	○ ○ ○○	
#	（Parameter）○	
*	○ ○○ ○	&
=	○ ○ ○ ○ ○	,
+	○ ○ ○	+

O, it is used a same code with program number O.

"#" of EIA code must be set with parameter.

But, it can not be employed without holes; the Latin alphabet can be employed, if the # is also used, **Note that, its original meaning has not to be applied!**

Parameter number

0	3	1	7									

Leading hole

User macro program A uses the especial word, see the following:

OR, XOR, IF, GOTO, EQ, NE, GT, LT, GE, LE.

User macro program B uses especial additional word, see the following:

AND, SIN, COS, TAN, ATAN, SQRT, ABS, BIN, BCD, ROUND, FIX, FUP, WHILE, DO, END.

2.10.10 Limiting Proceeding

(1) Variables

#0, #1～#33, #100～#149, #500～#509. System variable.

(2) Available variable values

The max. value $\pm 10^{47}$, the min. value $\pm 10^{-29}$.

(3) The effective constant value in <Expression>.

The max. value ±99999999 the min. value ±0.0000001.

Decimal point: available

(4) Operational accuracy: decimal system 8 digits

(5) Macro program call nestification level: the max. four layers

a) Repeated recognition number: 1~3

b) Nestification of [ ]: the max. five layers

c) The nestification level of subprogram call: up to 4 layers

d) The limit of user macro program A, and user macro program B fits with all the above items, the user macro program can be only fitted the following items.

(i) The variable other than system variable can be employed.

(ii) The following operations can be performed among variables: +, -, OR and XOR.

(iii) IF [<Conditional>] GOTO n can be transferred

(iv) It can be simply call and modal call.

2.10.11 P/S Alarm Explanation

(1) Alarm number 004

Fit address does not found in the actual position.

(For example)

X1*1，

Word 1 of X1 occurs after reading, the next address occurs, the number 004 alarm issues when the power * occurs.

(2) Alarm number 114

The formula depart from <Formula> is incorrect, this alarm occurs in the following circumstance.

(a) The word following the address is digit, the formula other than 0, -, #, [and +.

(For example) XF1000;

XSIN[10]；

(b) IF (WHILE) [<Formula> △△ <Formula>] inexistence.

（例） [IF#1 EQ #2 GOTO 10；

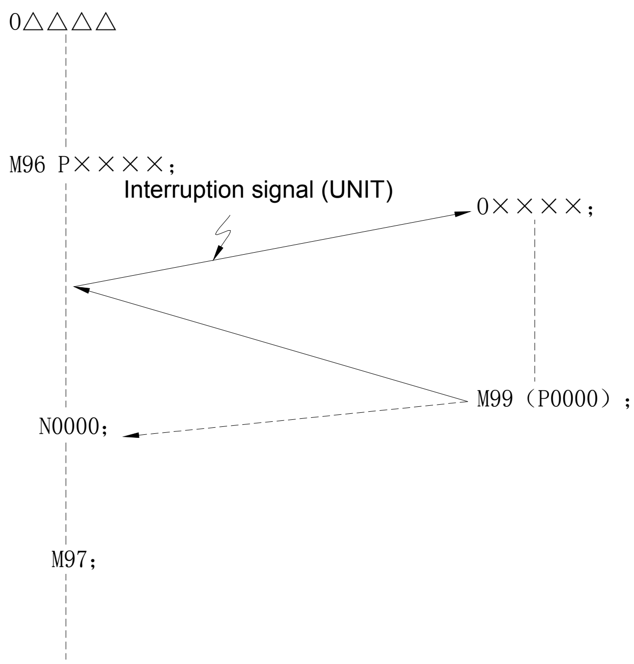
(For example) [IF#1 EQ #2 GOTO 10;

WHILE[#1 SIN #2] DO1；

WHILE [#1 SIN #2] DO1;

2.10.12 Macro Program Interruption Function (Macro Program B)

If macro program interruption function performs among M96Pxxxx; M97 and blocks, an interruption signal is entered to the NC in the midway, the control turns to Pxxxx program.



Setting M99, program returns to the original program by macro program interruption service program, its sequence can be set with address P.

Note 1: For macro function, refer to the appendix 12 for details.

Note 2: It is very necessary to refer to the appropriate manual issued by the machine tool builder when using this function.

2.10.13 External Output Command

When the standard user macro program command is entered, the following macro program command can be performed (This is regarded as external output command).

（a） BPRNT

（b） DPRNT

（c） POPEN

（d） PCLOS

The commands are reaching the object for variable value and words output by RS232 interface

These commands are specified in terms of the following sequence.

(1) Open command: POPEN

It is very important to deal with the external I/O device and the interface before inputting a series of data commands.

(2) Data output command: BPRNT and DPRNT

Data output command is required

(3) Close command: PCLOS

This command is employed when all data output command are ended. The external I/O device is departed from interface.

2.10.13.1 Open Command POPEN

This command should be offered when external I/O device is jointed with the treated interface before entering a series of data commands. The control code of DC2 output from NC side.

2.10.13.2 Data Output Command BPRNT，DPRNT

Data output command BPRNT, DPRNT

（1） BPRNT [a #b [c] ……]

The digit of significant below the decimal point

Character Variable

Character output and variable binary output are performed when performing BPRNT command.

(a) Word, the specified words are output in terms of ISO code, the command word can be performed is as below shown:

Letters (A~Z)

Numbers

Special character (*, /, +, -)

An asterisk (*) is output by a space code

(b) All variables are stored with a decimal point. Specify a variable followed by the number of significant decimal places enclosed in bracket. A variable value is treated as 2-word (32-bit) data, including the decimal digits. It is output as binary data staring from the highest bytes.

(c) When specified data has been output, an EOB code is output in terms of the setting code ISO.

(d) Null variables are regarded as 0.

（2） DPRNT [a #b [c d] ……]

Number of significant decimal places

Number of significant digits in the integer part

Variable

Character

The DPRNT command outputs characters and each digit in the value of a variable in terms of the code set in the settings (ISO).

(a) For an explanation of DPRNT command, see items (a), (c) and (d) for BPRNT command.

(b) When outputting the variable, specify # followed by the variable number, then specify the number of digits in the integer part and the number of decimal places enclosed in bracket.

One code is output for each of specified number of digits, starting with the highest digit. For each digit, a code is output according to the settings (ISO). The decimal point is also output using a code set in the settings (ISO).

Each variable must be a numeric value consisting of up to eight digits. When high-order digits are zeros, these zeros are not output if PRT (bit 7 of parameter 315) is 1. If parameter PRT is 0, a space

code is output each time a zero is encountered.

When the number of decimal places is not zero, digits in the decimal part are always output. If the number of decimal places is zero, no decimal point is output.

When PRT (bit 7 of parameter 315) is 0, a space code is output to indicate a positive number instead of +; if parameter PRT is 1, no code is output.

2.10.13.3 PCLOS

PCLOS：

In order to release the connection of I/O unit, specify this command when all data output commands have terminated. DC4 control code is output from the NC

2.10.13.4 Required Setting When Using this Function

(1) The punch output is applied using the output unit by setting the number 341 parameter, but it can not be input the storage box in this occasion.

(2) The number of output unit is preset to the number 341 parameter in terms of the above items, and the reading and each data of punching interface are set for number 310~313 parameters. (Baud rate etc)

(3) ISO code is set to output code.

(4) Setting number 315 parameter to determine when DPRNT command inputs data, whether outputting the space to the previous 0.

0	3	1	5		PRT							
					7	6	5	4	3	2	1	0

During the data outputting, the leading zero is treated by PRT DPRNT command, see the following:

0: Output space 1: not output

2.10.13.5 Notes

1) The POPEN, data output (BPRNT, DPRNT) and close commands (PCLOS) are not needed serially set. When the open command is set at the beginning of program, till the open command is not need set any more before the close command is set.

2) Setting the paired open and close command without omission.

In other word, the close command sends out at the end of program. Open command is not applied and the close command can not be set correspondingly.

3) The data output command in program is stopped using reset operation, and the following data is erased.

Correspondingly, if the resetting treatment is set with M30 or the other similar commands at the end of data output program. The close command is applied at the end of this program, it is

necessary to wait a moment before staring M30 or the other resetting treatment, till all data are output.

4) This function needs to select macro program B and I/O interface.

2.11　Tool Lifetime Management

2.11.1　Overview

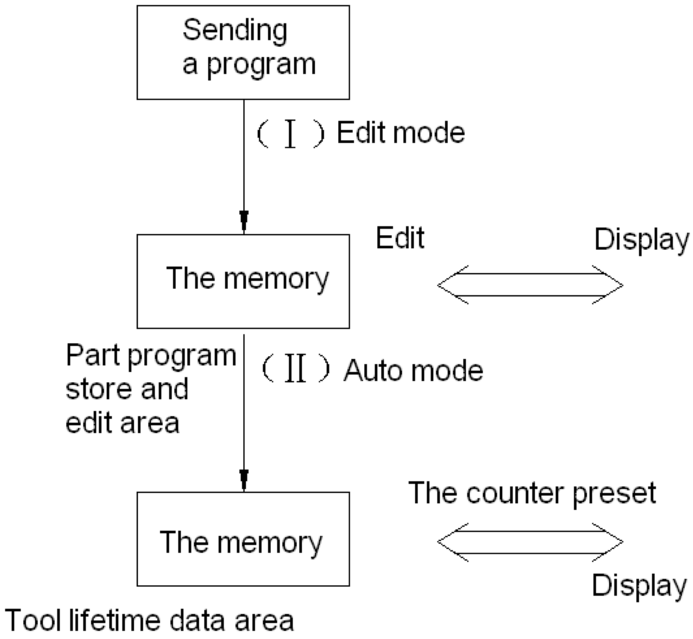
The tools are divided into many groups, and the tool lifetime is specified in each group by time or frequency. The tool lifetime management is the accumulative tool lifetime of each group and the tool is changed based on the specified sequence in the same group.

2.11.2　Setting the Tool Group

The tool sequence and the tool lifetime in each group are preset in NC device based on the following format.

Format	Meaning
O□□□□	Program number
G10L3；	Start setting the tool group
P□□L△△△△；	The digits after P is the group number (1~16). The digits after L is the tool lifetime（1～9999）(Note 3).
T△△□□；	（1） △△is the tool number.
T△△□□；	（2） □□ is the offset number. Select the tool based on the sequence （1）→（2）→…→（N）
⋮	
⋮	
T△△□□；	（N）
P□□L△△△△；	Setting the data of the next group.
T△△□□；	
⋮	
G11；	Setting the tool group end
M02（M30）；	Program end

Chapter 2　Programming



The setting operation is as below:

（i）The edited tool lifetime management program is sent to NC. The program is registered into the part program memory for display and editing.

（ii）In Auto mode, the cycle start is operated to run the program. The data is saved into the tool lifetime data area of the memory, meanwhile, the previous registered tool lifetime data of each group are all cleared. And the tool lifetime counter is also cleared. Once the data are registered, they will not be cleared even the power supply is cut off.

The following is the practical example of the format:

00001;	
G10L3;	
P01L0150;	
T1101;	
T0203;	Setting data of group 1
T0504;	
P02L0100;	
T1205;	
T1307;	Setting data of group 2
T0915;	
T0712;	
P03L0050;	
T0808;	Setting data of group 3
T1514;	
G11;	
M02;	

Note 1: The tool group number specified by P may not be consecutive, and the tools of all group numbers are not necessary to be set.

Note 2: The same tool number T can occur for many times in the setting data.

Note 3: The tool lifetime is displayed by frequency or time (minute) is set by the parameters（309—LCTM）**.**

Note 4: The registered group number is 16 and 16 tools are saved in each group, so totally 256 tools can be saved.

Note 5: In one machining process, for one tool with more than two offset numbers, the setting should be operated as below:

Format	Meaning
⋮ P04L0500; T0101; T0105; T0203; T0206; T0202; T0304; T0309; P05L1200; T0405; ⋮	The tool of group 4 can be used for 500 times (minute) based on the sequence of （1）→（2）→（3）. (1) In one machining process, the same group is specified for 3 times, the offset number sequence is as below: The tool corresponding to (1): 01→05→05， (2) The tool corresponding to (2): 03→06→02， (3) The tool corresponding to (3): 04→09→09.

2.11.3 The Tool Group Number Specified in the Machining Process

In the machining process, the following T codes are specified the tool group number:

Format	Meaning
⋮ ⋮ T△△99； └── group number ⋮ T△△88	Until now, all the tools are used up, the tools of △△ group are started to use. 99 means to differ the common commands. The tools of △△ group are cancelled, 88 means to differ the common commands.

The following is the practical example:

Format	Meaning
T0199；	The previous tools end, the tools of 01 group are started to use.
T0188；	The tool offset of 01 group is cancelled.
T0508；	The tools of 01 group end. The tool number 05 and the offset number 08 are selected.
T0500；	The # 05 tool offset is cancelled.
T0299；	The #5 tool ends, and the tool of group 02 is started to use.
T0199；	The tools of group 2 end and the tools of group 01 are started to use. If the group commands many offset numbers, #2 offset number is selected. Otherwise, the previous same offset number is used.

2.11.4 Executing the Tool Lifetime Management

2.11.4.1 The Counter of the Tool Lifetime

（1） The lifetime is specified by frequency

From the cycle start of the machining program to M02 or M03, NC is in the resetting state; at this period, the tool group counter for one machining program should be added up 1. No matter how many times the same group number is specified in one machining program, the counter is only added up 1, the maximum lifetime is 9999 times.

（2） The lifetime is specified by time (minute)

In this case, the machining process is executed from the specified T△△99 to the other tools specified by T code. In the cutting mode, the tool actual used time is counted once in 4 seconds except for the time of single block stop, feed hold, rapid feed and dwell. The maximum time can be set for 4300 minutes.

No matter in whatever situation, the tool lifetime counter of each group is counting. The content of the calculator won't be cleared at power-off.

Note: When the lifetime is specified with the frequency, M02 or M30 is executed, the external resetting signal is input or resetting is performed and the rewind (REW) signal is sent to NC.

2.11.4.2 The Tool Change Signal and the Tool Change Resetting Signal

After one tool lifetime ends, the other tool is selected based on the specified sequence. When the last tool lifetime of one group is also used up, the tool change signal is output. After changing the tools of the group, it is displayed on LCD screen. The corresponding group number is specified and

the tool change resetting signal is input or MDI/LCD panel is operated (refer to 2.11.5.3) to clear all the data of the tool group. Like the lifetime counter: *, @ etc (refer to 2.11.52) are cleared at the end of the tool lifetime. When the tool change resetting signal occurs, the tool change signal is automatically released. After the machining is restored, the 1st tool is started to be selected from the group.

Note: When the tool lifetime is specified by the time, once the lifetime is used up, the tool change signal is also output even in the machining period, while the machining continues till it ends. For the frequency, even the tool lifetime ends, the tool change signal is output after M02 or M03 is reset.

2.11.4.3 Tool Skip Signal

If one tool lifetime isn't used up, the tool is completed to be changed with one of the following methods.

(i) Specify the group in which the tool is, and input the tool skip signal. Use the next T code to command the next tool in the group.

(ii) Not specify the group number, and input the tool skip signal. It's assumed to select the present tool, others are same as（i） item.

（i）or (ii) is set by parameter（309-TLSK）. The lifetime is started from 0. However, the tool skip signal is input into the last tool, the tool change signal is output.

2.11.4.4 New Tool Selection Signal

In one group, a new tool is going to be selected. When T code of the tool is output, the signal of selecting new tool is also selected at the same time. When the new tool is selected, the signal can be used to automatic measure the tool compensation amount.

Note: When STL or SPL is ON, the tool change resetting signal and the tool skip signal can not be input.

2.11.5 Display and Inputting the Tool Data

2.11.5.1 Display and Rewriting the Tool Group Number

In the part program memory and edit areas, the tool group number can be displayed and rewritten like the common program. Definitely, the program is rewritten as 11.2; otherwise, it can't be saved in the tool lifetime area.

2.11.5.2 Display the Tool Lifetime Data during the Machining Program Executing

In any mode, press the diagnosis key twice, display the 1st page of the tool lifetime data on LCD screen.

Display the data of two groups on one page. Press the page key to display the data of each group one by one. The tool change signal of maximum 5 groups can be displayed at the bottom of each page. If 6 or more groups exist, one arrow (cursor) is displayed in LCD. To understand the data of one group, select the address N. Input the group number and press the input key, or move the cursor to GRP of the next group and display the data.

2.11.5.3 Preset the Tool Lifetime Counter

Select MDI mode to rewrite the tool lifetime counter.

(i) Input P□□□□ and press the input button

Then, the counter of the group, in which the cursor is, is preset as □□□□, while the other data of the

group remains unchanged.

(ii) Input P-9999 and press input button

All the execution data of the group in which the cursor is, including * are completely cleared, which is same as the function of the tool setting (refer to 2.11.3).

2.11.6 Setting the Parameter

The parameters of the tool lifetime management are set as below:

3	0	9	TLSK	GRST		LCTM				

LCTM =1: The tool lifetime is specified by time.

=0: The tool lifetime is specified by frequency.

GRST =1: After the resetting signal is output, the execution data of each group is cleared.

=0: After the resetting signal is output, the execution data of the lifetime end group is cleared.

TLSK =1: Skip the tool and the group number is input.

=0: Skip the tool while the group number is not input.

2.11.7 Alarm

The following alarms are not GSK983T standard ones.

Alarm No.	Meaning
140	The group number exceeds the maximum value 16.
141	The group number specified in the machining group isn't set.
142	The tool number in the group exceeds 16.
143	T code doesn't exist in the program setting the tool group.
145	The group number specified by T△△88 doesn't comply with the current group number.
146	At the starting of the program, P and L are not specified.
147	The tool group number exceeds the maximum value 16.

2.11.8 Other Precautions

In the last part, the part program memory and edit areas are reduced some space for the tool lifetime data area. In EDIT mode, it will occupy more space when the data are registered into the part program and the edit area introduced in 11.2.

Chapter 3 CNC Specification and Configuration Function

<table>
<tr><th>Item</th><th>Name</th><th>Specification</th></tr>
<tr><td>1</td><td>Controlled axis</td><td>X and Z axes, two axes</td></tr>
<tr><td>2</td><td>Controlled axis number meanwhile</td><td>Two axis linkage control meanwhile</td></tr>
<tr><td>3</td><td>Language</td><td>The standard language is Chinese; it can be set as English.</td></tr>
<tr><td>4</td><td>PLC</td><td>PLC MODEL B is as follows:
<table><tr><td>Number of input points</td><td>192 points</td></tr><tr><td>Number of output points</td><td>128 points</td></tr><tr><td>Program</td><td>Max. 5000 steps</td></tr></table></td></tr>
<tr><td>5</td><td>Environmental conditions</td><td>（2） Environment temperature
0℃～45℃ during running, -20℃～55℃ in other situations
（3） Relative humidity
≤90%（Without condensation） ,≤95%（40℃）
（4） Vibration
<0.5G during running, <1G in other situations
（5） Environmental air
Please contact with the manufacturer when NC device is used in the environment with much dust, cutting oil solution or organic solution in high concentration.</td></tr>
<tr><td>6</td><td>Self-diagnosis function</td><td>(1) Servo system
a. The system alarms when the error of the error register is greater than the setting value in the stop state.
b. The system alarms when the value of the error register is greater than the maximum setting value.
c. The system alarms when the position detection system is abnormal.
d. The system alarms when the drifting voltage is too big.
e. The system alarms when the speed control unit is abnormal.
(2) NC
a. The system alarms when the memory is abnormal.
b. The system alarms when ROM and RAM are abnormal.
c. The system alarms when the microprocessor is abnormal.
(3) On-line state
a. Display NC state on LCD.
b. Display I/O state on LCD.
c. Display PLC state on LCD.</td></tr>
</table>

<table>
<tr><th>Item</th><th>Name</th><th>Specification</th></tr>
<tr><td>7</td><td>Setting unit</td><td><table><tr><td>Min. input unit</td><td>X and Z axes</td><td>0.001mm</td><td>0.0001inch</td></tr><tr><td>Min. movement</td><td>X axis</td><td>0.0005mm</td><td>0.00005inch</td></tr></table></td></tr>
</table>

		unit | Z axis | 0.001mm | 0.0001inch Here, X axis is the diameter programming (the radius programming can be selected), the minimum input increment can be input in 0.01mm in metric system, which are set by parameters.
8	Max. commanded value	±99999.999mm ±9999.9999inch
9	Input format	Adopt the changeable programmable field, character and address format
10	Decimal input	The values with the decimal point can be input. The addresses X，Z，U，W，I，K，R，E and F can be with the decimal point.
11	Rapid feed rate	The maximum rapid feedrate override can reach 60，000mm/min or 2400 inch/min (set by parameters). The rapid feedrate override can be rewritten into F0, F25%（selectable）, 50%, 100%.
12	Spindle speed function (S2 digits)	The spindle speed can be specified by two digits after address S.
13	Spindle speed function（S4 digits）	The spindle speed (the standard configuration is set before dispatch) can be specified by four digits after address S.
14	Tool function (T2 digits/ T4 digits)	The tool selection and the tool offset are specified by two or four digits after address T.
15	Miscellaneous function (M2 digits)	The ON/OFF signal on the machine side can be controlled by two digits after address M. One block can only be commanded one M code.
16	Automatic acceleration/deceleration	During rapid feeding, the linear acceleration/deceleration in Jog or Auto mode is adopted to save the positioning time.
17	External power supply ON/OFF	Except for power supply ON-OFF button on NC device, the contact signal from the machine operation panel out of NC device can also cut off the power supply.
18	Emergency stop	Use the emergency stop to stop all of the feed commands, and the machine immediately stops.
19	External resetting signal	NC resetting can be operated out of NC device. All feeding commands stop through resetting. The machine decelerates and stops.
20	Feed hold	The feeding of all axes temporarily stops, restarting can be operated by the cycle start key, and the manual operation can be executed before refeeding.
21	Servo ready signal	After the servo system is ready, the signal is sent from the machine side. The brake axis is locked without sending the signal. NO READY is displayed on LCD without the signal.
22	NC ready signal	When NC power supply is connected and controllable, the signal is sent into the machine side. When the power supply is cut off and the control unit is too hot, stop sending the signal to the machine side.

23	NC alarm signal	The signal is sent when NC is in the alarm state.
24	Distribution completed signal	When the movement command is executed completely, CNC sends the signal. If M, S, T and movement commands are in one program block, the signal can execute M, S and T function after the movement commands end.
25	Signal in the cycle running	The signal is sent when NC is in cycle running.
26	Automatical running start indicator signal	The signal is sent when the automatic running is started.
27	Feed hold indicator signal	The output signal is sent when the feed hold is in the dwell state.
28	Interlock	The commanded axis feeding can be forbidden independently. Any one axis is interlocked in the movement, the interlock signal is released after all axes in the mechanical part decelerates and stops, and then the axes are accelerated and restarted.
29	Overtravel	The signal occurs after the machine reaches the stroke end, the axis movement decelerates and stops, the overtravel alarm occurs.
30	Miscellaneous function lock	BCD code signals of M, S, T and B function and the strobe signal are forbidden to send to the machine side.
31	Machine lock	The machine does not move, while the coordinate is displayed the movement. The machine lock is also valid during the machine block executing.
32	Dry run	In dry run mode, the feedrate is JOG one. The rapid feed command (G00) remains unchanged; the rapid feed (select) is valid. However, the dry is also valid based on the parameter setting.
33	Single block	Commands can be executed in the programs one by one.
34	Skip optional block	Through connecting the optional block skip switch on the operation panel, the block with slash code at the beginning can be ignored.
35	The added memory program number	Based on the number of standard programs, add 96 programs, totally 191 programs (require to store the part program and edit C～F).
36	Sequence number search	The sequence number in the present program can be indexed on MDI/LCD panel.
37	Program number search	The sequence number of the 4 digits after O can be indexed on MDI/LCD panel, for ISO code, colon (:) can replace O.
38	Program restart	Specify the sequence number and restart from the program.
39	Program key lock	Key lock the programs 9000～9899, display, setting and editing are forbidden.
40	Butter register	Pre-read the next block to eliminate the interruption of NC command movement due to the reading time. At the right bottom of LCD screen, BUF is displayed when the data are input into the buffer register.
41	Screw pitch error compensation in memory type	Compensate the errors caused by the mechanical wearing of the thread, which can improve the machining precision and extend the mechanical lifetime. And the compensation data are saved in the memory as the parameters, which can

		omit the block and setting procedures.
42	Backlash compensation	The mechanical wearing compensation is executed on the machine side. Each axis can independently take the minimum movement unit as the unit, and the compensation amount can be set as the parameters in the range of 0～255.
43	Absolute/incremental command	Specify the absolute command with addresses X and Z, the incremental one with U, W. These address words can be specified in one block.
44	External data input	The data are sent to NC from the machine outside, and the specified operation is executed. The types corresponding to the external data are as follows: （1） External workpiece number search C （2） External tool compensation C （3） External alarm information （4） External operator information
45	Tool offset	One lowest valid digit in T code (T code of two digits) or the lowest two valid digits (T codes of four digits) specify the tool offset number. The offset value corresponding to the offset number must input from MDI key and register into the memory. The offset compensation of 32 groups is standard, and 64 groups can be selected.
46	Manual absolute ON/OFF	Whether the distance of the tool is moved by manual operation is added to the coordinates can be selected by turning the manual absolute switch on or off on the machine operation panel. Manual absolute switch ON: Add OFF: Not add.
47	Override cancel	The cutting feedrate can be fixed in 100% through the signal from the machine side.
48	Manual continuous feed	（1） JOG feeding JOG feedrate can be switched in 24 levels with the rotary switch; the ratio is in 24 equal levels. And the standard panel is divided into 20 levels. （2） Manual rapid feeding The rapid feeding can be operated in Jog mode, the rapid feedrate is set by parameters.
49	Incremental feed	The following incremental feeding amount can be positioned, so the manual position can be executed in high efficiency. （Incremental feeding amount） 0.001；0.01；0.1；1；10；100mm (Input in metric system) 0.0001；0.001；0.01；0.1；1；10inch (Input in inch system)
50	Reference position return A	The reference position return A has the following contents: （1）Manual reference position return （2）Reference position return check (G27) （3）Automatic reference position return (G28) （4）Return from the reference position (G29)
51	Reference position return B	The reference position return B not only has the function of the reference position return A, but also has the 2nd reference position return (G30).

52	Stored stroke limit 1	The stored stroke limit 1 is set by the parameter to divide the area outside which forbids the machine movement.
53	Stored stroke limit 2,3 (G22，G23)	(1) The stored stroke limit 2 The area inside is the forbidden one, which is set by the parameter. (2) The stored stroke limit 3 The area inside or outside is the forbidden one, which is set by the parameter or the program. Whether the stored stroke limits 2, 3 are valid or invalid is set by G codes. G22: valid; G23: invalid.
54	Automatic coordinate system setting	The workpiece coordinate system is set when the reference position return in Jog mode.
55	Workpiece coordinate system movement	The coordinate system movement is executed when the coordinate system doesn't comply with the actual one during programming.
56	Cutting feedrate（G98，G99）	The cutting feedrate is classified into the feeding per revolution and feeding per minute, which is set by G codes. G98: Range of feeding per minute (mm/min，inch/min）: 1 m/min～30,000m/min 0.01 inch/min～12000.00 inch/min G99: Range of feeding per revolution (mm/r, inch/r） 0.01 mm/r～500.00mm/r 0.0001inch/r～50.0000inch/r The speed can be selected from 0 to 200%, and 10% be one level.
57	Setting the coordinate system S（G50）	Use the commanded values of X and Z axes after G50, and the present tool coordinate value can be taken as the coordinate system of the commanded value (About the details, refer to 2.5.10).
58	Positioning (G00)	Each axis can feed independently through commanding G00. The axis decelerates and stops at the end position, whether the in-position check (whether the machine reaches the commanded position) is executed is set by the parameters (About the details, refer to 2.5.1).
59	Linear interpolation（G01）	The linear interpolation can be commanded by G01 on the feedrate specified by F (About the details, refer to 2.5.2).
60	Arc interpolation（G02，G03）	Programming can be executed in the range of 0º～360º (About the details, refer to 2.5.5). G02: Clockwise（CW） G03: Counter-clockwise（CCW）
61	Dwell (G04)	The dwell time of the next block movement can be specified by G04 (About the details, refer to 2.5.9). It is commanded by address P or X.
62	Conversion between inch/metric system（G20，G21）	The input of inch or metric system is selected by G code (About the details, refer to 2.5.11). G20: Inch input G21: Metric input
63	Thread cutting (G32)	The straight, taper and end face threads of the lead are specified by F code, and the cutting are commanded by G32.

64	Cutter compensation (G40~G42)	The cutter compensation is specified by G codes (G41 or G42), the tool nose radius value of each tool can be stored into the memory. G40: Cancel the offset G41: Compensation in the left of the tool movement direction G42: Compensation in the right of the tool movement direction The maximum compensation value is ±999.999mm or ±99.9999inch. (About the details, refer to 2.6.3).
65	Canned cycle of sole type (G90，G92，G94)	There are three types of the canned cycle (About the details, refer to 2.7.1.): （1）Cutting cycle A (G90)……cutting of outer diameter （2）Thread cutting cycle （G92） （3）Cutting cycle B……for the turning end face.
66	Compound canned cycle（G70~G76）	There are 7 types of the compound canned cycle (About the details, refer to 2.7.2.): （1）Finishing cycle (G70) （2）Outer diameter roughing cycle（G71） （3）End face roughing cycle (G72) （4）Closed-loop cutting cycle (G73) （5）End face cutting groove cycle (G74) （6）Outer diameter groove cycle (G75) （7）Thread cutting cycle (G76)
67	The constant circle speed control A, B (G96, G97)	The function can change the revolving speed of the spindle motor, so the tool can remain the constant circle speed in any position. The constant circle speed can be directly specified by S4 digits. Whether the constant circle speed is valid or invalid is represented by the following G codes: G96: Valid G97: Invalid In the constant circle speed control A, the signal of 12 bits in binary system without the contact, which is the calculated spindle speed, is sent to the machine. In the constant circle speed control B, the analog signal in the direct ratio with the spindle speed is sent to the machine.
68	Thread cutting specified by E code of 6 digits	F code is replaced by E code to specify the precise thread lead in the following range: Metric input: 0.0001 mm ~500.0000mm Inch input: 0.000001 inch ~9.99999inch
69	Changeable thread lead cutting	Based on the added or subtracted value of the thread lead per revolution, the thread cutting of the changeable lead can be operated.
70	Dwell during the thread cutting	When the thread cutting is executed by G92 or G96, the tool retracts immediately and returns to the start position if the system is in feed hold state.

71	Consecutive thread cutting	The consecutive thread cutting blocks can be specified by G32, G34, and the pulse distribution interruption time is between blocks. To make the interruption time 0, the cutting depth is changed continuously, and the thread cutting is executed in one point repeatedly, and the cutting won't damage the thread teeth.
72	Manual insert function	During the automatic operation, when the tool has slight feeding or retraction, rotate the manual pulse generator during machining, the tool moves the corresponding distance based on the pulse number generated by MPG. In Auto mode, add up with the commanded pulse; however, the workpiece coordinate system has been moved the corresponding pulse distance.
73	Display the running time	Each unit s, min, h or 0.1 h can be displayed and reset.
74	Automatic acceleration/ deceleration during cutting feed	During the cutting feed or the manual continuous feeding, the time constant of the automatic acceleration/deceleration function with the index type is set by the parameter from 8ms～4000ms.
75	special G code C	Refer to 2G code list in Appendix.
76	The incremental offset input	The original set tool offset value is added one input tool compensation value.
77	Automatic tool compensation n	The tool is moved to the measured point, and NC corrects the present set tool compensation value.
78	Direct input the measured value of the offset	The tool offset value can be set by measuring the workpiece diameter and the length after manual cutting. The value is input in MDI.
79	Input the offset value	The standard tool moves to the fixed point of the machine in Manual mode, and the measured tool moves to the same point on the machine. After pressing U Input and W Input, the tool offset values of X and Z axes are respectively set.
80	Tool lifetime management	The tools of the turret tool post can be classfied into some groups. The tool lifetime (usage time or frequency) of each group is set. Each tool is selected by the specified group number. And the tool is used one time, the usage time or frequency is counted. When the set lifetime is used up, the next tool in the same group is automatically selected. When the lifetime of all tools ends, one signal is output to inform the operator to change the tools.
81	Tool figure and tool wearing offset	The tool offset is classified into the tool shape or the tool installation position offset (the tool shape offset) and the tool nose wearing offset (the tool wearing offset), which can be respectively set. This characteristic greatly simplify the machining of the machine tool. In other words, the operator can only set and improve the wearing amount (small value).
82	External workpiece number search A	The workpiece number is selected by the switch on the machine and the corresponding program is found and executed.
83	Radius programming in X axis	When the X axis movement amount is input by the radius value, the diameter value movement along X axis can't be input.
84	X axis mirror image	The symbol commanded by MDI can reverse in X axis, and is operated by the switch on the machine control panel.

85	Mirror image of double tool post	The symbol of X axis value in the program can be changed by G codes to realize the symmetrical cutting (mirror image).
86	Angle chamfering and corner R	For the straight cutting vertical to or parallel with the axis, the angle chamfering or corner R can be inserted. The data of angle chamfering or corner should be specified by addresses I, K and R.
87	Specified arc radius R	During the arc interpolation, the arc radius can be directly specified instead of addresses I and K, the arc radius must be equal to or less than 180°.
88	Feedrate sine curve control	For the arc cutting command, I axis does not move on arc plane, only the interpolation (the assumed axis) can control the feedrate of the other axis, which forms the change of the sine curve.
89	Skip function	In the linear interpolation, one signal is sent from outside, the remaining movement amount can stop and transfer to the next block.
90	Additional optional block skip	At the beginning of the block, the numerical values 1~9 can be specified. 9 optional block skip switches are installed on the machine. When the optional block switch n is connected, the block with /n is skipped.
91	Custom macro A, B	The machine manufacturer and the user can design the special function, there are two types A and B due to the function limit.
92	Menu switch	The ON/OFF function can be operated on MDI/LCD to replace the switches on the machine panel. Therefore, the quantity of switches on machine operation panel can be reduced.
93	Figure dimension direct programming	The straight angle, angle chamfering and corner values on the machining diagram can be directly programmed. Further, any angle, angle chamfering and corer can be inserted between two straight lines.
94	Sequence number comparison and stop	During executing the program, once the block with the sequence number occurs, which is same as the preset number, the system enters the single block stop state the block is completely executed.
95	The path display	The tool path of the program can be described on LCD. The machining situation can be checked by observing the path displayed on LCD.
96	The external position display	The position displayer of each independent axis can be installed on the machine.
97	Manual pulse generator	The manual pulse generator installed on the machine operation panel is used for inching feeding of the machine. 100 pulses per revolution are sent from the MPG. The movement distance of per grid is switched with the switches on the machine side. Metric input: 0.001, 0.01, 0.1mm; Inch input: 0.0001, 0.001, 0.01inch;

Chapter 4　Operation

4.1 Power ON/OFF

4.1.1　Power On

（1）　Please connect the machine power supply based on the manual of the machine manufacturer.

（2）　Connect CNC power supply after pressing the power on button 

on the machine operation panel.

4.1.2　Power Off

（1）　Confirm the cycle start button 

LED indicator on the machine operation panel is OFF.

（2）　Confirm the machine movable part is in the stop state.

（3）　Press the power off button 

on the operation panel to cut off CNC power supply.

（4）　About cutting machine power supply, refer to the manual of the machine manufacturer.

Note: After the system powers off, it should be powered on, again after at least 5 seconds.

4.2 Program Lock Switch

Normally, only the program lock is ON, the program can be edited.

Notes:

① The program lock state can be checked on the program check interface.

② Some parameter rewritten items can only be executed when the program lock is ON.

③ Whether some functions can be executed is set by the system NC parameter 10.6 when they are OFF.

4.3　Operation of the Machine Operation Panel

4.3.1　Operation Panel

The function of operation panel and the allocation of switches may vary as the different machines; about the details, please refer to the machine manual and PLC manual with corresponding version.

The manual only introduces the operation panel with one standard configuration, which is shown as the following figure.

4.3.2 Emergency Stop

In emergency, press this button, all axes movement of the machine stops immediately, and the button keeps self-lock in the stopping position.

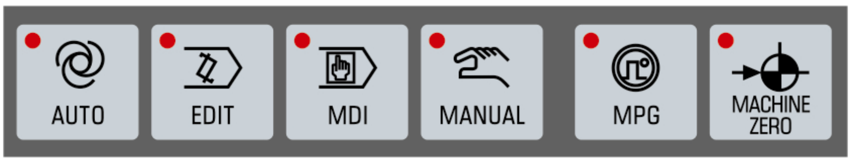
Emergency stop button

The button release button varies as the different machine manufacturer, and is released through CCW or CW rotation.

Notes:

1. The machine should be operated zero return, again after releasing the button.
2. The troubleshooting should be executed after releasing the button.
3. The motor enable is cut off after pressing the button.

4.3.3 Mode selection



Operation mode is selected:

Mode	Function
Auto	（1） The programs in the memory are executed. （2） The sequence number can be searched for the program in the memory.
Edit	The following program editing can be executed:

	（1） The program is saved in the memory. （2） The program rewriting, inserting and deleting are executed. （3） The programs are output in the memory.
Manual data input (MDI)	The manual data input operation can be realized in MDI/LCD panel.
Manual	The manual feeding can be executed.
MPG	MPG feeding can be executed.
Machine zero return	X and Z axes zero return function can be executed.

4.3.4 Operation Relative to the Manual

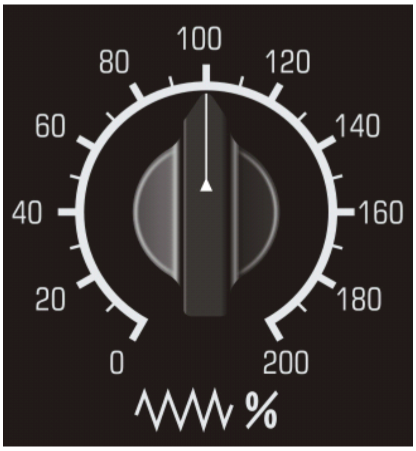
The following manual operation can be executed with the switches and buttons on the operation panel and MPG.

4.3.4.1 Manual Continuous Feeding

The manual continuous feeding can make the machine move.

（1）Select "manual" operation mode.

（2）Select the movement feedrate (the feedrate override switch can only be valid when the rapid movement switch is OFF.)



Manual feedrate mm /min

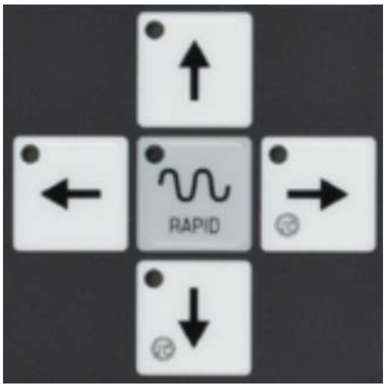
Rotary switch position	Inching feedrate			
	Metric thread		Inch thread	
	mm /min	Inch /min	Inch /min	mm /min
0	0	0	0	0
1	1.0	0.04	0.02	0.508
2	1.4	0.055	0.028	0.711
3	2.0	0.079	0.04	1.02
4	2.7	0.106	0.054	1.37
5	3.7	0.146	0.074	1.88
6	5.2	0.205	0.104	2.64
7	7.2	0.283	0.144	3.66
8	10	0.394	0.2	5.08
9	14	0.551	0.28	7.11
10	20	0.787	0.40	10.2
11	27	1.06	0.54	13.7
12	37	1.46	0.74	18.8

13	52	2.05	1.04	26.4
14	72	2.83	1.44	36.6
15	100	3.94	2.00	50.8
16	140	5.51	2.80	71.1
17	200	7.87	4.00	102
18	270	10.6	5.40	137
19	370	14.6	7.40	188
20	520	20.5	10.40	264

Note 1: The numerical values are different on the above list based on the machine manufacturer.
Note 2: The error of the listed feedrate is about ±3%.

（3）Selecting the movement axis

Press the direction buttons corresponding to the movement axes, and the tool moves along the selected direction.



Note 1: Two axes can be controlled meanwhile in Manual mode.
Note 2: After connecting the power supply, even the "mode selection" switch is preset as the "manual mode" and one axis is selected, the tool does not move along the selected axis, so the axis should be selected, again.

（4）Rapid movement

Press the button, the machine moves along the preset axis at the rapid traverse rate.

Note 1: The manual rapid traverse time constant and the acceleration/deceleration mode is same as those when G00 is commanded, which is set by the machine manufacturer.
Note 2: The manual rapid traverse rate can be set as 0.25, 0.5 and 1 time of the G00 commanded speed.F0 is set by the system parameter 113, which is set by the machine manufacturer.
Note 3: With the stroke limit selection function in the memory type (parameter #9.0　ISOT) and the axis with the reference position return, after power on or the emergency stop, the feedrate will not change into the rapid feedrate after pressing the rapid feed button as long as the reference position return isn't executed. Because the stroke limit in the memory type does not function before the manual reference position return, it can prevent the movement is executed along the axial direction at rapid rate to reach the stroke end position.

4.3.4.2　MPG

The machine movement and the feedrate can be accurately adjusted through MPG.

（1）　Select MPG.

（2）　Select the movement amount.

X1 means the movement amount times 1; X10 means the movement amount times 10; X100 means the movement amount times 100.

Input	Movement amount each grid		
	×1	×10	×100
Metric input	0.001mm	0.01mm	0.1mm
Inch input	0.0001inch	0.001inch	0.01inoh

（3） Select the movement axis: Press X or Z axis movement key (+ positive or –negative is OK).

（4） The hand wheel for rotating the MPG

CW rotation…… + direction

CCW rotation……- direction

（The directions are different based on the specification of the machine manufacturer.）

Note 1: If MPG is rotated at the speed more than 5 rev/sec, the differential value occurs between the MPG revolving amount and the machine movement distance, so the MPG rotation speed should not be too fast.

Note 2: When the machine is with the MPG, the incremental feeding function is not used.

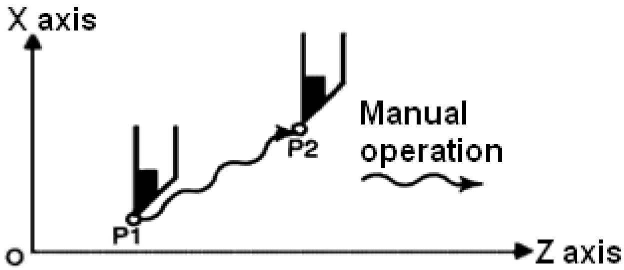
Note 3: When ×100 override is selected, the rapid revolving MPG makes the tools of the machine or the worktable is moved based on the speed of "rapid movement". Then, the machine gets impact if it stops suddenly. Select the automatic acceleration and deceleration function is also valid and it can reduce the mechanical impact.

Note 4: Select the content with the MPG insert function, refer to 4.3.7.

4.3.4.3 Manual Absolute ON/OFF

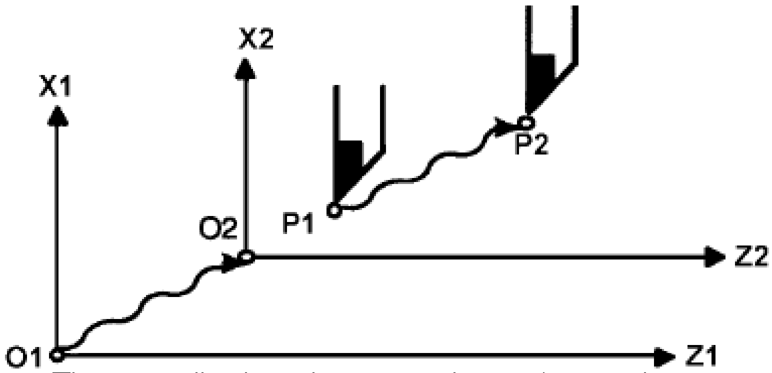
Whether the distance the tool is moved by manual operation is added to the automatic running coordinates can be selected by turning the manual absolute switch on or off on the display panel. When the switch is turned on, the distance the tool is moved by manual operation is added to the coordinates. When the switch is turned off, the distance the tool is moved by manual operation is added not to the coordinates.

（1） When the manual absolute switch is "ON", the coordinates change by the amount of the manual operation.



The coordinate values are changed
by the amount of the manual operation.

（2）　When the manual absolute switch is "OFF", the coordinates do not change by the amount of the manual operation.



The coordinate values remain unchanged.

The following describes the relation between manual operation and the coordinate values when the manual absolute switch is turned on or off, using a program example.

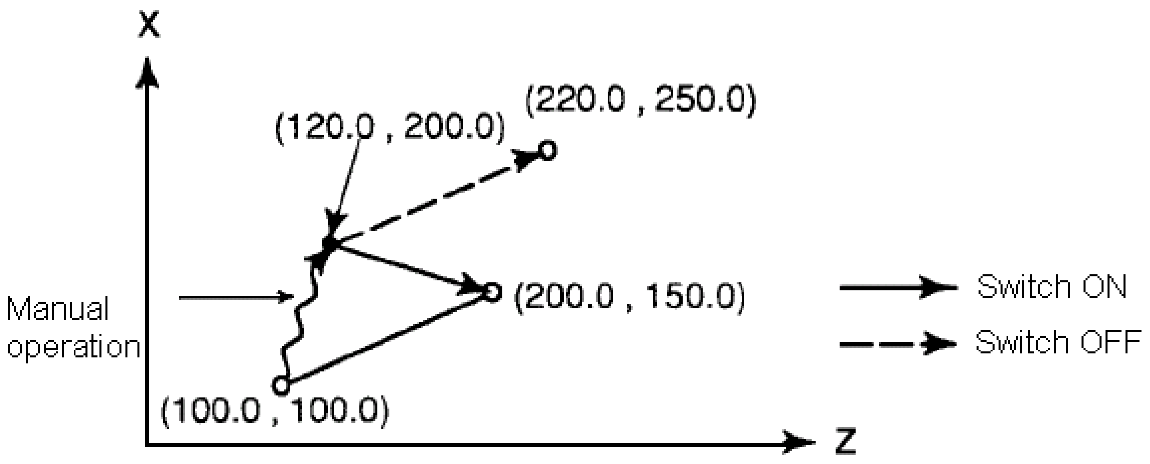
G01 X100.0 Z100.0 F100；　(1)

X200.0Z150 ；　(2)

X300.0Z200.0 ；　(3)

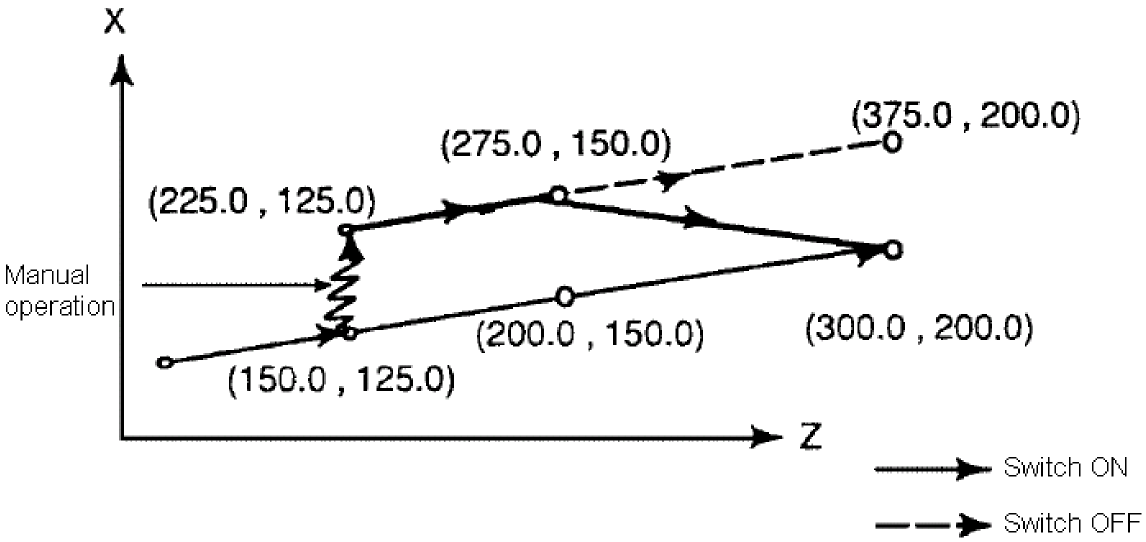
（A）　Manual operation after the end of the block

Coordinates when block (2) has been executed after manual operation (X axis +20.0, Z axis +100.0) at the end of movement of block (1).



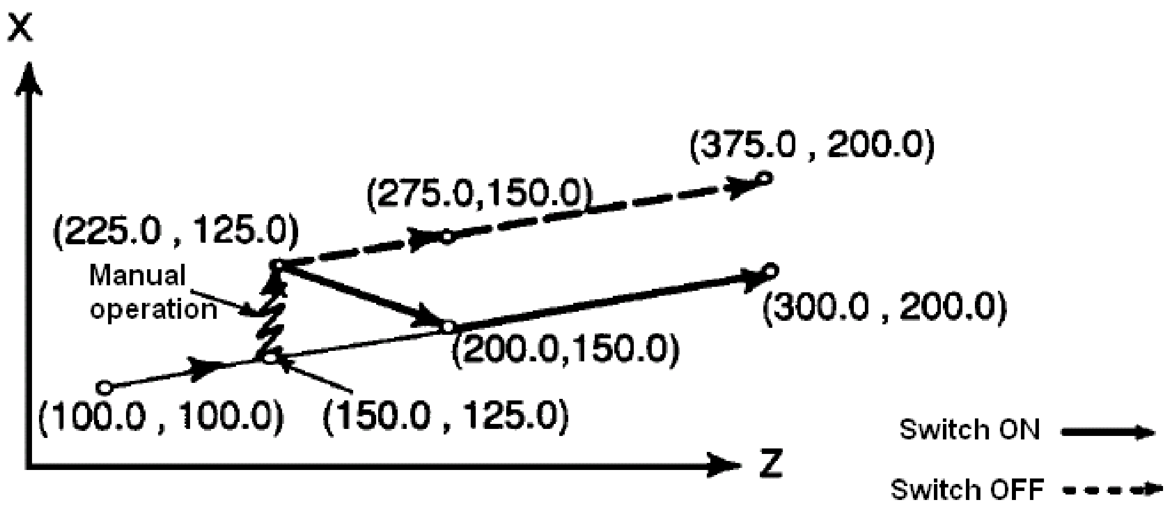
（B） Manual operation after a feed hold

Coordinates when the feed hold button is pressed while block (2) is being executed, manual operation (X axis +75.0) is executed, and the cycle start button is pressed.



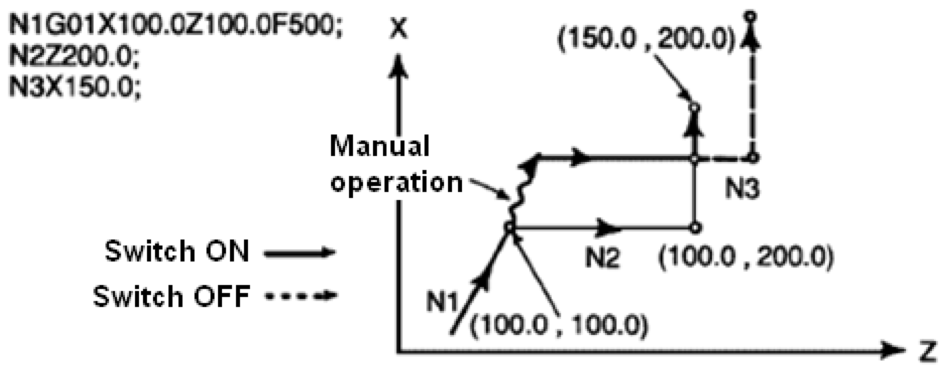
（C）When reset after a manual operation following a feed hold

Coordinates when the feed hold button is pressed while block (2) is being executed, manual operation (X axis +75.0) is performed, the control unit is reset with RESET button, and block (2) is read again.



（D）When a movement command in the next block is only one axis

When there is only one axis is commanded, only the commanded axis returns.



（E）Manual operation during the cutter compensation (when the next move block is an incremental.)

When the following commands are incremental ones, operation is same no matter when the switch is ON or OFF.

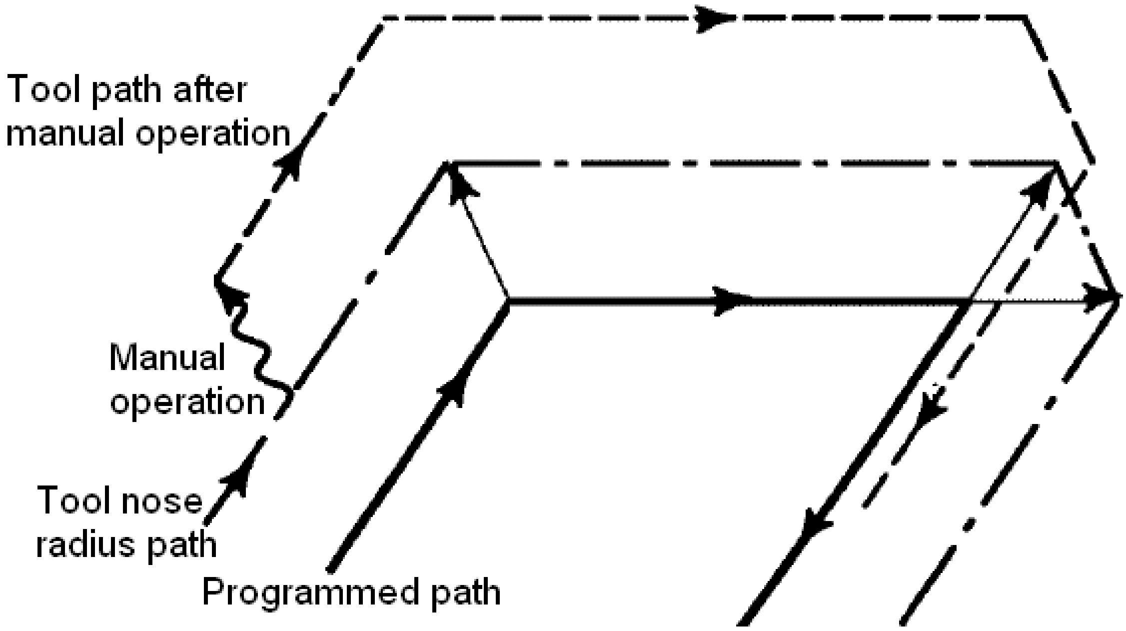
（F）Manual operation during cutter compensation

When the switch is “OFF”:

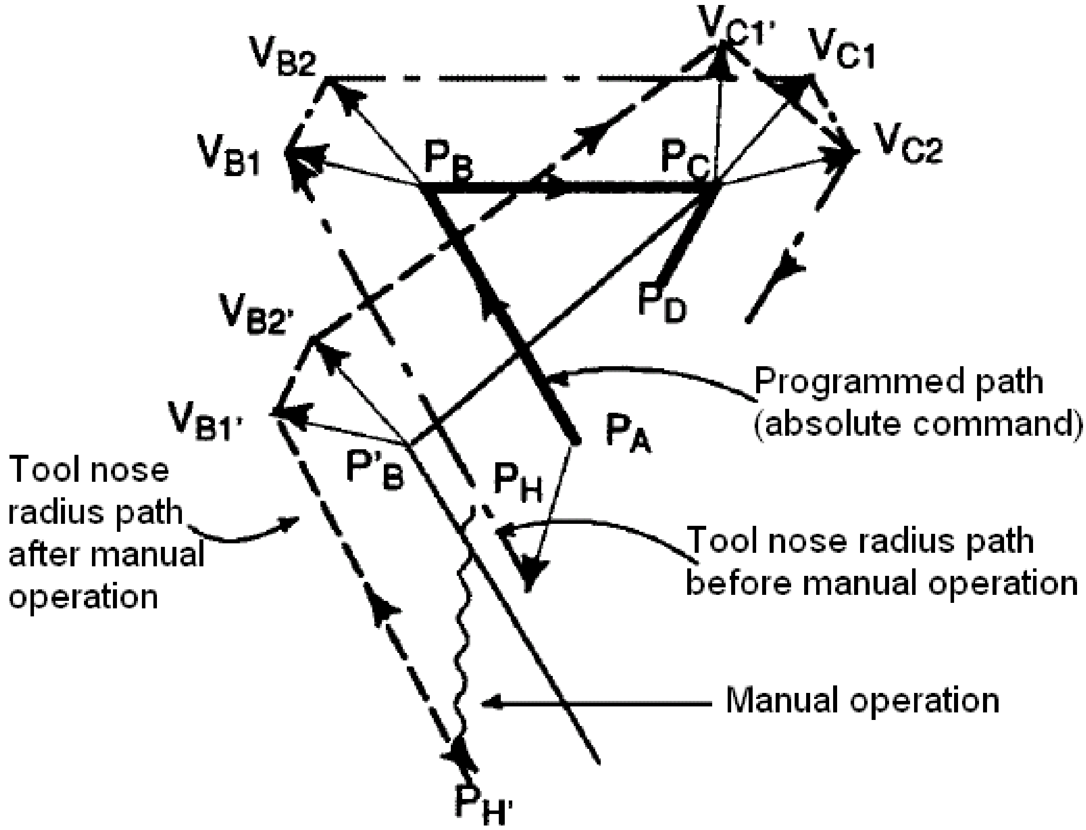
After manual operation is performed with the switch OFF during cutter compensation, automatic operation is restarted then the tool moves parallel to the movement that would have been performed if manual movement had not been performed. The amount of separation equals to the one that was performed manually.

When the switch is “ON”:

Operation of the machine upon return to automatic operation after manual insert with the switch is ON during execution with an absolute command program in the cutter compensation mode will be described. The vector created from the remaining part of the current block and the beginning of the next block is shifted in parallel. A new vector is created based on the next block, the block following the next block and the amount of manual movement. This also applies when manual operation is performed during cornering.



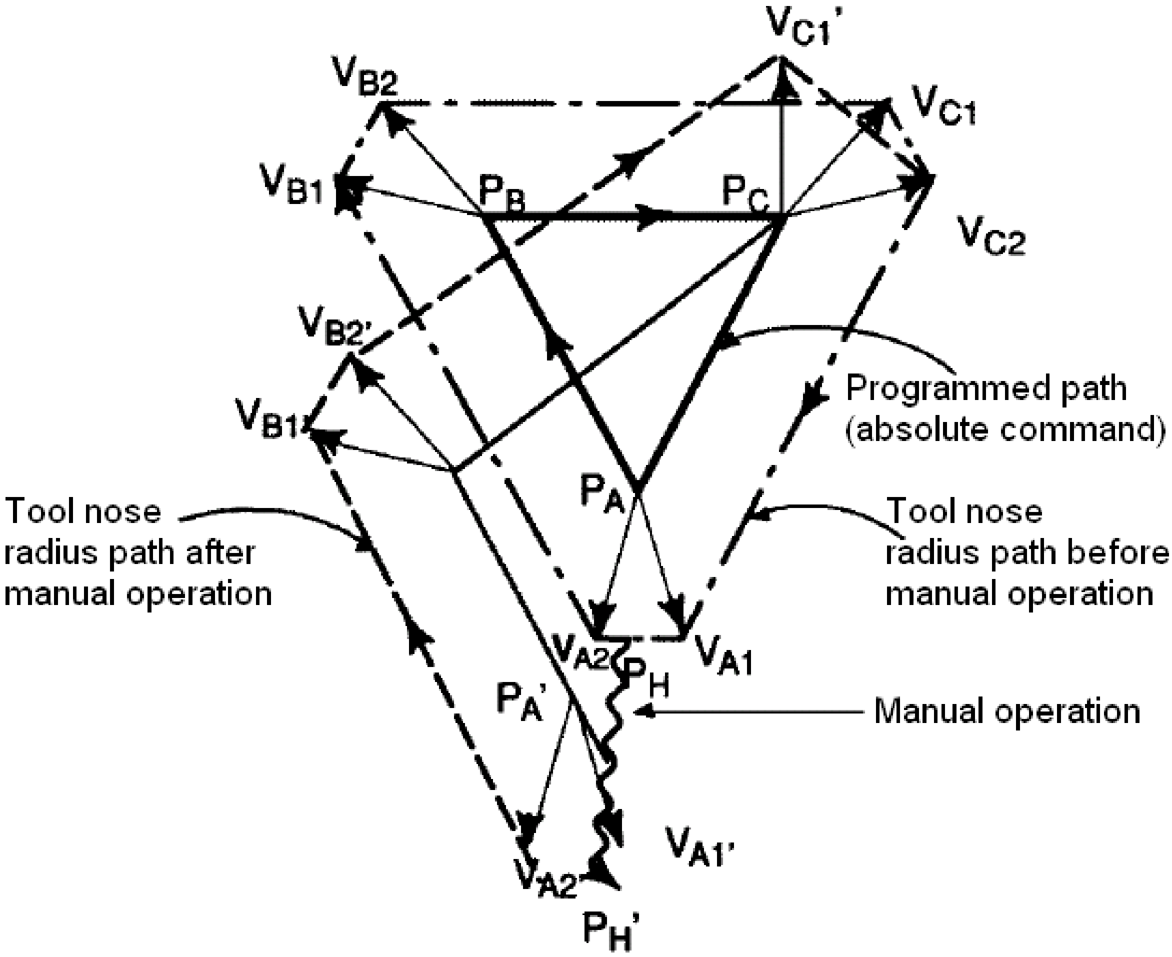
Manual operation performed when in other than cornering



Assume that the feed hold was applied at point P_H while moving from P_A to P_B of programmed path P_A, P_B and P_C and that the tool was manually moved to P_H'. The block end point P_B moves to the point P_B', and vectors V_{B1} and V_{B2} also move to $V_{B1}'$and V_{B2}'. Vectors V_{C1} and V_{C2} between the next two blocks P_B-P_C and P_C-P_D don't compensate and new vectors $V_{C1}'$and V_{C2}'(V_{C2}'=V_{C2} in this example) are produced from the relation between P_B'—P_C and P_C-P_D. However, since V_{B2}' is not a newly calculated

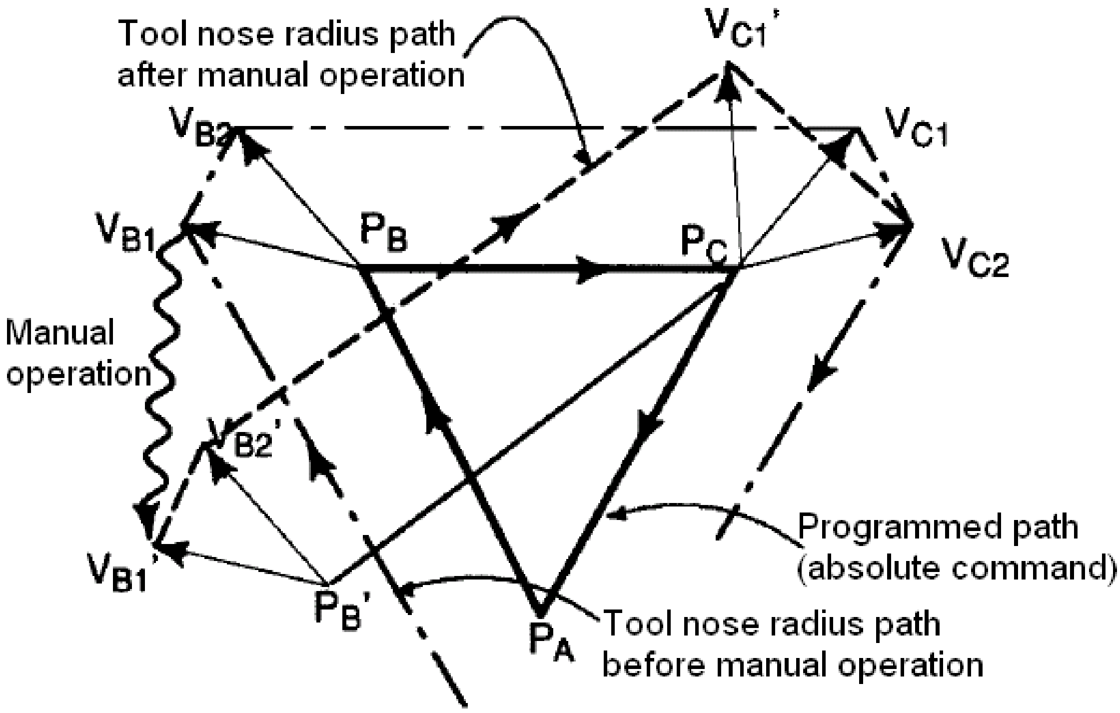
vector, correct offset is not performed at block P_B'—P_C. Offset is correctly performed after P_C .

Manual operation during cornering



This is an example when manual operation is performed during cornering. $V_{A2'}$, $V_{B1'}$ and $V_{B2'}$ are vectors moved in parallel with V_{A2}, V_{B1} and V_{B2} by the amount of manual movement. The new vectors are calculated from V_{C1} and V_{C2} . Then correct cutter compensation is performed for the blocks following Pc.

Manual operation after single block stop



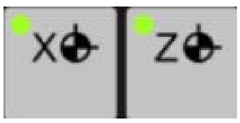
Manual operation was performed when execution of a block was terminated by single block stop. Vectors V_{B1} and V_{B2} are shifted by the amount of manual movement. Sub-sequent processing is the same as the case described above. An MDI operation can also be intervened as well as manual operation. The movement is the same as that by manual operation.

4.3.5 Manual Reference Position Return

（1）Press the zero return 

button (after pressing the button, the left top corner LED lamp is ON).

（2） The machine enters the zero return self-protection state after pressing the zero return axis direction keys.

(3) After each axis zero return completes, the two buttons 

LED lamps on the left top corner are ON.

Note: If the wrong reference position return direction is selected, the movement is not executed.

4.3.6 Operation about Automatic Running

The machine automatically runs based on the program commands. The automatic running operation steps are as below:

4.3.6.1 Start of Automatic Running

The steps of starting to run the program stored in the memory:

（Ⅰ）Select the program number, refer to “program number search” in Section 4.4.14.

（Ⅱ）Select the Auto mode.

（Ⅲ）Press cycle start button.

When the button of “cycle start” is pressed, the system starts automatic running, and “cycle start” button lamp is ON meanwhile.

Note: Press “cycle start” button is invalid or cancelled in the following situations:

(a) Press “feed hold” button.

（b）Press “emergency stop” button.

（c）The “resetting” signal is connected (About the details, please contact the machine manufacturer).

（d）“Mode selection” button is set in the wrong position (except for auto, edit and MDI modes).

（e）The sequence number is being searched.

（f）The alarm occurs.

（g）In automatic running (it is neither in feed hold nor in stop state).

（h）NC system is not ready.

4.3.6.2 Automatic Running Pause

（1） Press feed hold button.

Press feed hold button, the lamp of “cycle start” button is OFF, and the “feed hold” button lamp is ON. Then:

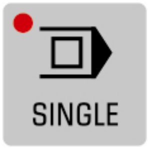
（Ⅰ）If the tool is being moved, the feeding stops after deceleration.

（Ⅱ）If it is “G04 dwell” state, the pause state won’t continue.

（Ⅲ）The machine running stops after execution of M, S or T function ends.

Note: When G32, G34, G76 or G92 (The situations of special G code B: G33, G34, G76 or G78; the situations of special G codes: G33, G34, G78 or G21) are used for thread cutting, even “feed hold” button is pressed, the feeding doesn’t stop until the command execution is completed. However, during the thread cutting cycle, press feed hold button, the tool is retracted in oblique line and returned to the cycle start position.

4.3.6.3 Single Block



When the button is pressed, press cycle start button, and then the control device stops after one block is executed each time.

（1）Reference position return and the single block command.

The reference position commands G28, G29 and G30 are valid for the single block function in the intermediate position.

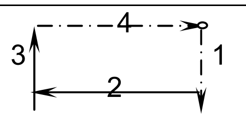
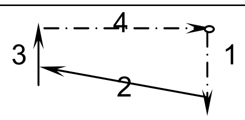
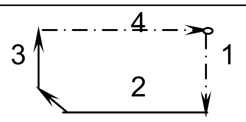
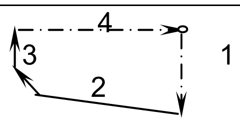
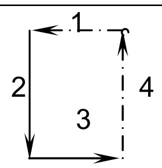
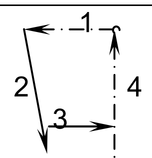
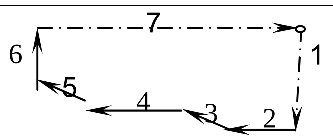
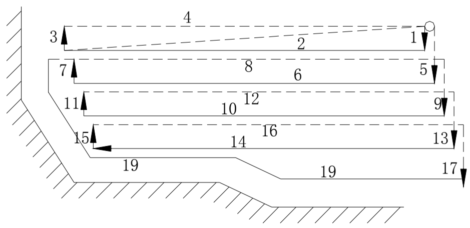
（2）Single block running in the canned cycle.

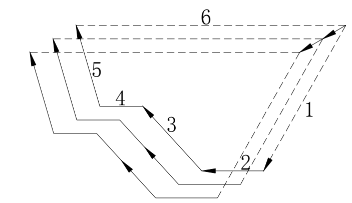
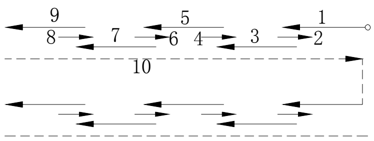
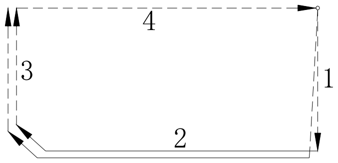
In the canned cycle, the single block stop positions are as below:

Note: If “single block” button is pressed, the actual running results of the canned cycles G90, G92, G94 or G70～G76 (in the special G code B: G77, G78, G79 and G70～G76；in the special G code C: G20, G21, G24 and G72～G78) are as below:

Rapid movement

Feeding

G Code			Tool path	Note
Standard	Special B	Special C		
G90	G77	G20	 	It’s assumed that the tool paths 1-4 are taken as one cycle, and it stops after the 4th path is executed.
G92	G78	G21	 	Same as above.
G94	G79	G24	 	Same as above.
G70	G70	G72		It’s assumed that the tool paths 1-7 are taken as one cycle, and it stops after each 7 paths are executed.
G71 G72	G71 G72	G73 G74	 **Note:** The path shown in the figure is the same as that of G71 and G72.	It’s assumed that the tool path from 1 to 4, from 5 to 6, 9 to 12, 13 to 16 and 17 to 20 are taken as one cycle, and it stops after each cycle ends.

G73	G73	G75		It's assumed that the tool paths 1-6 are taken as one cycle, and it stops after each 6 paths are executed.
G74 G75	G74 G75	G76 G77	 Note: The path shown in the figure is the same as that of G74 and G75.	It's assumed that the tool paths 1-10 are taken as one cycle, and it stops after each 10 paths are executed.
G76	G76	G78		It's assumed that the tool paths 1-4 are taken as one cycle, and it stops after each 4 paths are executed.

（3） Subprogram calling and the single block running

The single block doesn't stop in the block with commands M98P_, M99, G65, G66 or G67. However, if the addresses except for O, N or P are in the block, the single block stops even M98P_ or M99 command in the block.

4.3.6.4 Restarting after a Feed Hold or Stop

（1）Select " Auto" or " MDI" mode.

（2）Press cycle start button, the "cycle start" lamp is ON, while the "feed hold" lamp is OFF.

4.3.6.5 Manual Operation in Automatic Running

In automatic running, the automatic running pauses after the " feed hold" button or "single block" one are ON on the operation panel.

（1） Record the stop position coordinates shown by the position display unit.

（2） Manual operation is performed (refer to 4.3.4.3).

（3） The tool returns to the recorded stop position coordinates, which is the start position in the Manual mode.

（4） To restart the automatic running, restore the state of the work mode and the modal before the manual operation.

（5） Press cycle start button.

4.3.6.6 MDI Operation during the Automatic Running

（1） Press "single block" button, the single block is valid and stops after one block is executed.

（2） Select "MDI" mode.

（3） MDI operation is performed.

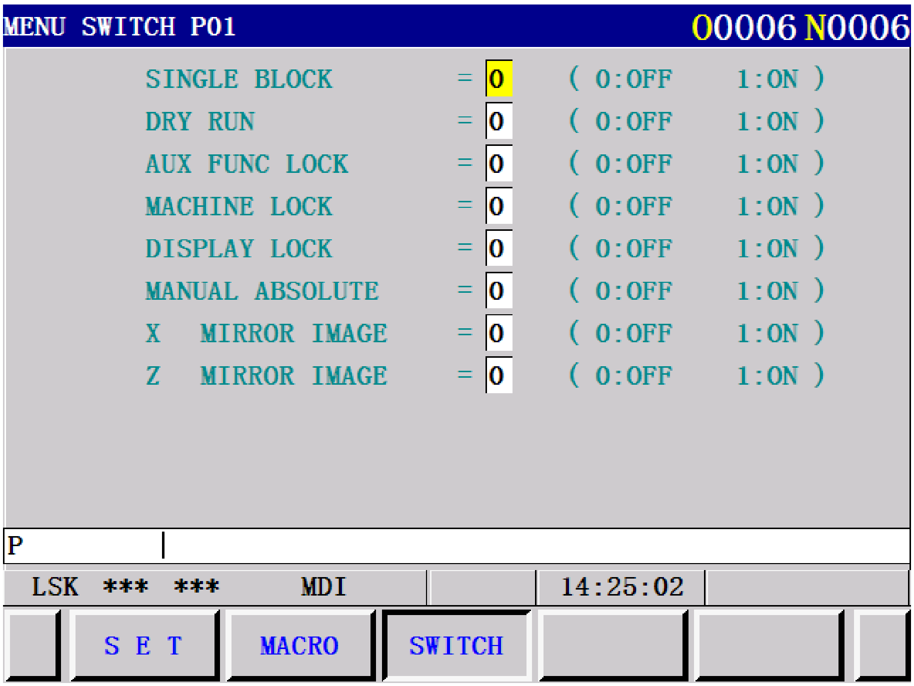
（4） To restart the automatic operation, select the original work mode and modal, press the cycle

start button on the operation panel.

Note 1: The MDI commands are affected by modal data before commanded automatic running.
Note 2: The MDI commanded modal data are still valid during the automatic running.
Note 3: The cutter compensation R is not executed in MDI operation.
Note 4: Press "feed hold" button during executing the next block, and the automatic running pauses; the automatic running can't restart during MDI command is operated.

4.3.6.7 Optional Block Skip

The function allows the block with the first word "/" is skipped.



At the beginning of the block, one slash with one numerical value (/n(n=1～9)) is specified and the BLOCK SKIP switch on the machine operation panel is ON, the specified optional block skip switch number n correspondence/the information included in n block are ignored in the memory mode.

When the BLOCK SKIP switch number n is OFF, the information of the block specified by /n is valid, which means the operator can decide whether skip the block with /n or not.

1 can be ignored in /1. However, when two or more optional block skip switch is used for one block, 1 in /1 can't be ignored.

Example

(Not correct) (Correct)

//3G100×10.0； /1/3G00×10.0；

The function is ignored when the program is registered into the memory, and the block with /n is also stored in the memory no matter whether the block skip switch is ON or OFF.

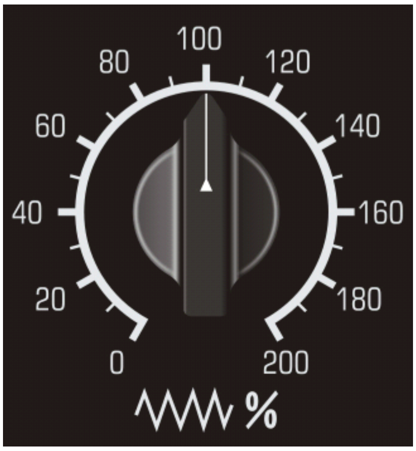
The stored program in the memory can be output no matter whether the block skip switch is ON or OFF. Even during the sequence number is being searched, the optional block skip switch is also valid.

The number of the optional block skip switch is different based on the different machines; about the details, refer to the manual of the machine manufacturer. Block skip switch on the operation

panel is normally set as the BLOCK SKIP 1.

4.3.6.8 Feedrate Override

The feedrate specified by the program can multiply by the override.



The commanded speed multiplies by the override corresponding to the scales of 0～200%, and the increment is 10%.

Note 1: The switch is used with the manual feedrate switch, which is set by the machine manufacturer.
Note 2: During the thread cutting, this switch is invalid and the override is normally 100%.

4.3.6.9 Dry Run

In the memory or in the cycle running commanded by MDI, the dry run is valid, F function is useless and the machine is moved at the following speed.

Rapid traverse button ON/OFF	Rapid traverse	Cutting feed
Rapid traverse button ON	Rapid traverse	Max. inch feedrate
Rapid traverse button OFF	Inch feedrate (refer to the notes)	Inch feedrate

Note 1:Whether the dry run is valid or not is set by NC parameter 6.0（RDRN）when the rapid feeding is commanded.
Note 2: Whether the dry run is valid or not is set by NC parameter 8.0（TDRN）during the thread cutting.

4.3.6.10 Machine lock

The movement command pulse stops when the screen（MACHINE LOCK）switch is ON or the operation panel machine lock is valid. Therefore, the feed cycle start or the position display in Manual mode continues to refresh based on the input commands, while the machine itself doesn't move and

the machine coordinate doesn't change, neither. The function is used for checking the program.

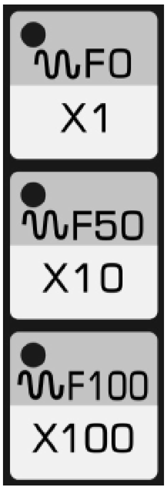
Note 1: When G27，G28 or G30 command is set, the machine won't move into the reference position, and the reference position return lamp is OFF.

Note 2: M, S or T functions are executed.

4.3.6.11 Miscellaneous Lock

When "MST" button is ON, the machine movable sheet iron is moved normally during the program running, while M, S and T function isn't executed, the function also applies to the program check.

4.3.6.12 Rapid Movement Override



On the machine operation panel, the rapid movement overrides buttons of the optional override 100%, 50% and F0 can be set. When the feedrate is 10m/min, the switch is set in the position of 50%, and the actual feedrate is changed into 5m/min. F0 is the fixed speed (feedrate) which is set by the machine manufacturer.

The function also applies to the following situations:

（1）The rapid movement speed of G00;

（2）The rapid movement speed of the canned cycle;

（3）The rapid movement speed of G27，G28，G29 and G30;

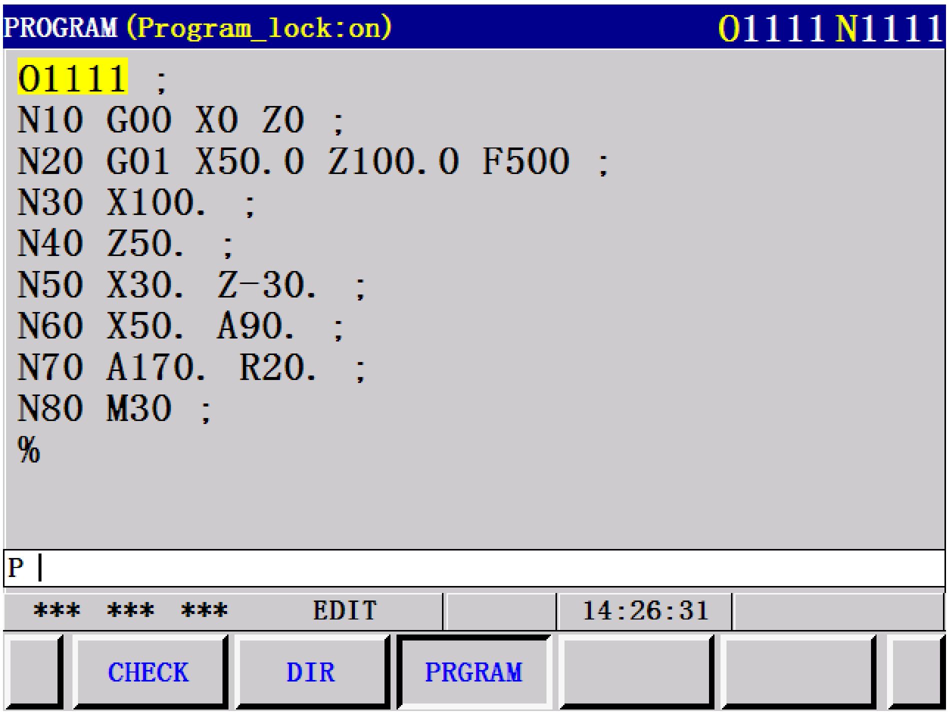
（4）Manual rapid movement speed;

（5）The rapid movement speed of the manual reference position return.

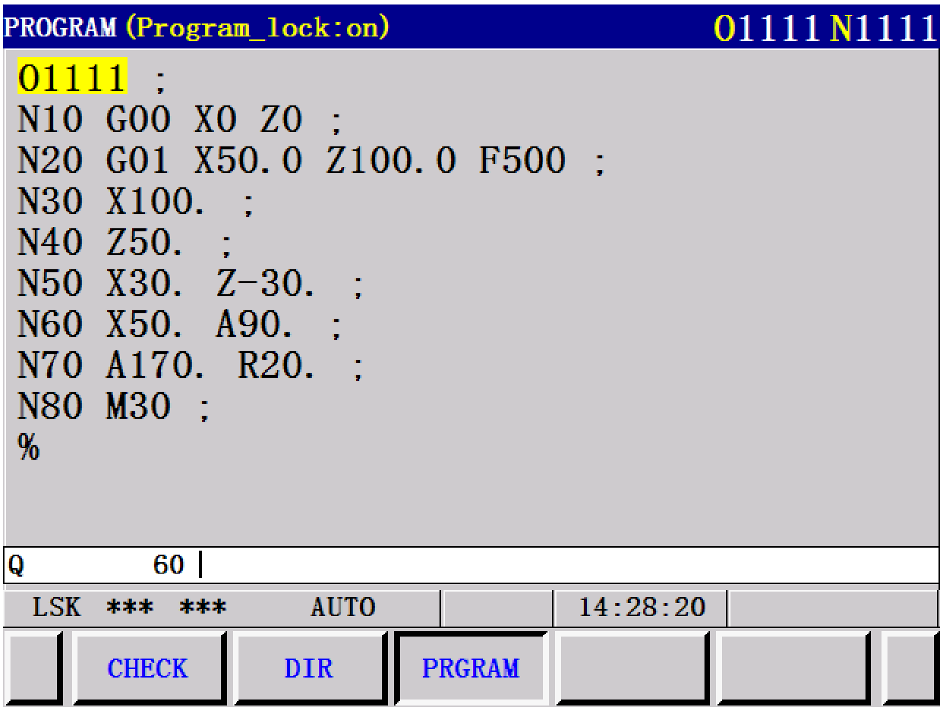
4.3.6.13 Program Restart

The function of the program restart is that when the tool gets damaged , the machining program is operated in the midway, or the program is interrupted for the other work, the program continues to execute from the interrupted position after getting ready, and the modal data before the interrupted position are all reserved, the following example is shown as below;

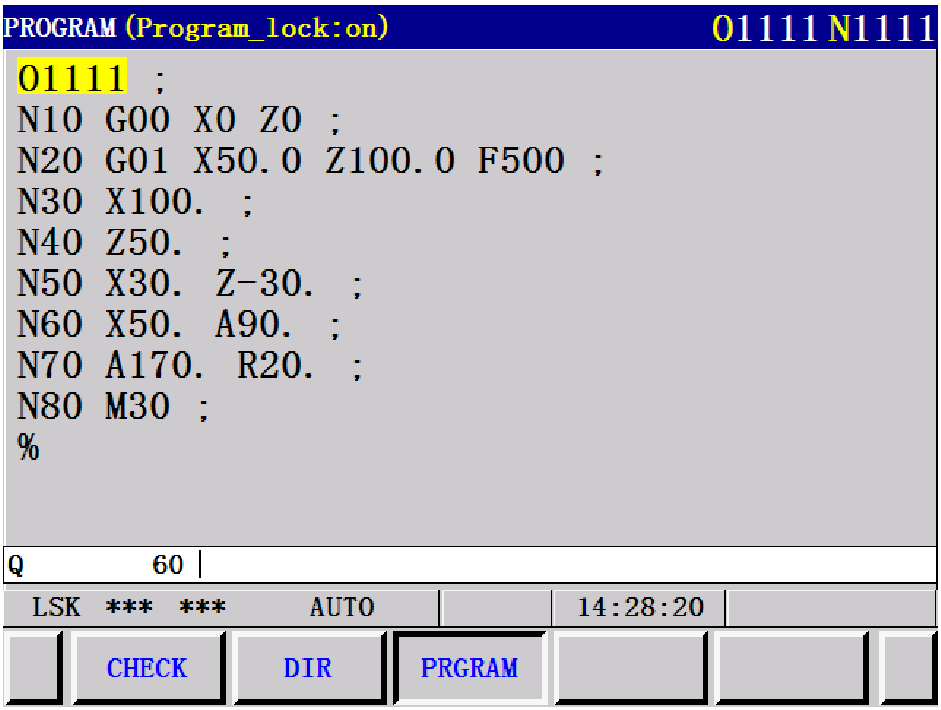
When the program O1111 runs to line N60, the program is interrupted to change the tool, and then the program continues to run from N60. The detailed operation step is as below:



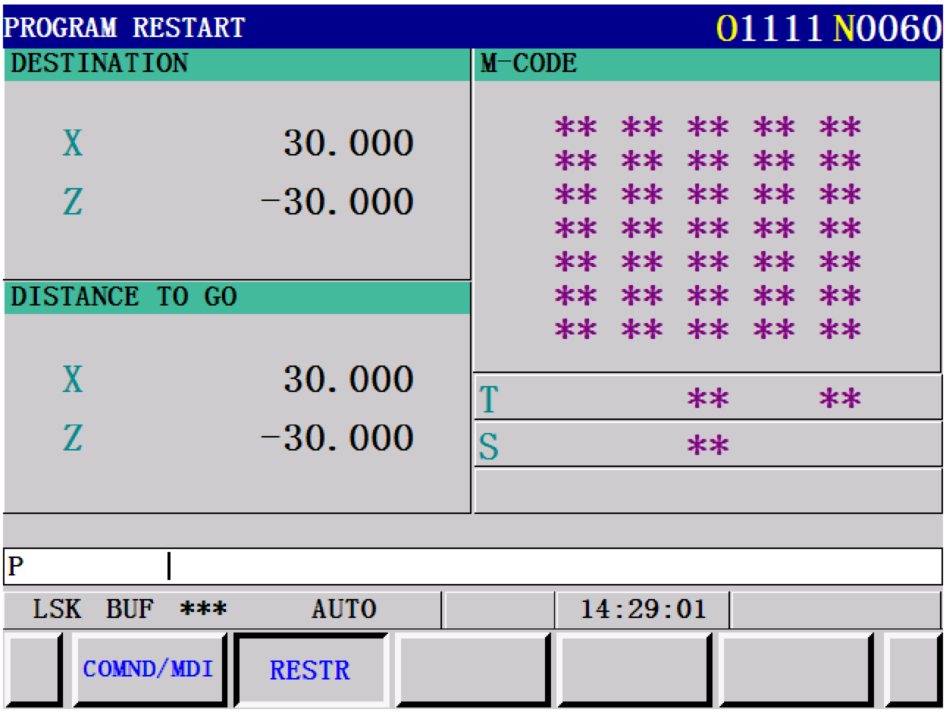
（1） In Auto mode, press the Program restart button on the operation panel, the lamp of the button is ON, and then input P (or Q) + the interrupted block number on the program interface, which is shown as the following figure:



（2） Input Q+ 60, and press the cursor ↓ key, search the sequence number N60 of the block, after N60 is searched, the interface immediately jumps to the program restart one, which is shown as below:



（3） Press program restart button, again on the operation panel, the lamp is OFF, and then switch into the program interface; press cycle start button, the program starts machining from the interrupted position.



Points for attention:

Note 1: Normally, P+program number (P mode) is selected, while Q+program number (Q mode) is selected only when the following situations occur.

（a） Cut off the power supply.
（b） Press the emergency stop button.
（c） The coordinate system changes after the last automatic operation is completed.

Note 2: In the following situations P sequence number , the cursor ↓, the program is not started.

（a） After the power supply is connected, the automatic operation is not executed.

(b) The automatic operation is performed after the emergency stop is released or after the stroke limit in memory type alarms.

(c) The automatic operation is performed after the coordinate system is set, changed or moved (the external workpiece zero offset amount changes).

The machining restart block is one of many blocks. The block is after the one of the last coordinate system is set or changed before the machining stops.

Note 3: In P and Q modes, the tool moves into the machining restart position with one axis each time. After the axis movement completes, the single block perhaps stops; while MDI operation can't be inserted but manual operation, and the returned axis can't move.

Note 4: During searching, when the movement signals and the offset amount, etc are different with the previous ones, the tool can't return to the previous restarting position of machining. Set the single block switch as ON or switch into the automatic mode to continue searching.

Note 5: During searching, when the feed hold remains valid or the resetting is operated after searching, the program should be restarted from the beginning. After the searching ends, parameter 007 "CLEAR bit" is rewritten into resetting state in MDI mode during resetting.

Note 6: When the program automatic start switch is ON, the cycle start can be ignored.

Note 7: The manual absolute switch should be always ON and the manual operation is executed no matter before or after the machining.

After the manual operation before resetting, one program restarting operation is commanded, or the manual operation is performed along with the axis without returning to the machining restarting position, it's assumed that the manual absolute switch is ON no matter whether the switch is ON or OFF.

Note 8: In the following situations, the tool can't return to the correct position in principle.

(a) The manual absolute switch is OFF during the manual operation.

(b) The tool is moved when the machine is locked or Z axis command is cancelled.

(c) Use the mirror image function.

(d) The coordinate system isn't set at the incremental programming starting position.

(e) The manual operation is inserted when the axis is operated the returning movement.

(f) After the program restart is commanded, the machine lock is released.

(g) The program restart command is sent when the program with the leapfrog cutting program or the block before the one with the absolute command is executed.

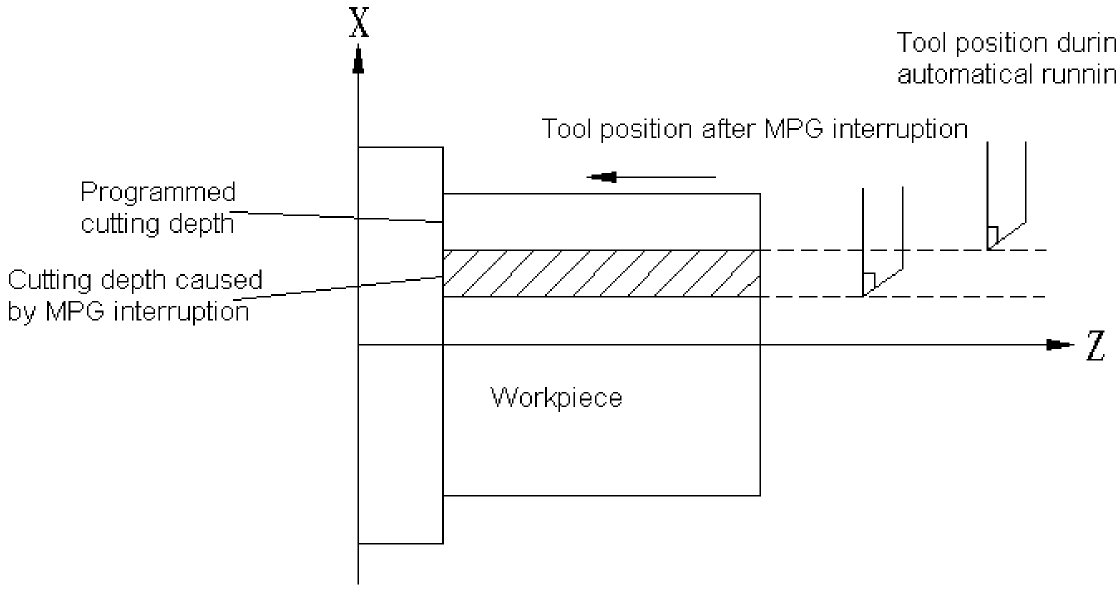
(h) When the coordinate system is set or moved after searching, while in the situation of (C), the tool return movement in P mode can be executed in the block in which the mirror image machining switch is OFF and in the following blocks. And in such situation, the mirror image machining state is same as that when the machining gets interrupted. No alarm occurs in any situation.

Note 9: The alarm NO.60 is issued when the specified block only includes M98, M99, the macro calling commands (M65, G66, G67) or the macro program sentence, or the program not specified is searched.

Note 10: After the power supply is connected, the emergency stop or the stroke limit alarm (stop immediately) is released; P/S alarm (98) occurs when the program restart is operated and G28 is detected while the reference position return isn't executed.

Note 11: After completion of search in program restart, P/S alarm (99) occurs when a move command is given with MDI before the axis movement operation.

4.3.7 MPG Interruption



4.3.7.1 Overview

The specified axis (set by parameters #314.0 and #314.1) can be moved by MPG manual operation during automatic operation, and added into the movement of automatic running.

4.3.7.2 Operation of MPG Interruption

In the following situations, MPG interruption can be operated by MPG.

（1） In automatic mode or MDI mode.

（2） Movement state: In linear, arc, helical or sine interpolation, MPG insert can be performed except for the following situations:

In the situations below, the manual interpolation can not be executed:

（Ⅰ）In the alarm;

（Ⅱ）The machine lock is valid;

（Ⅲ）The positioning is valid;

（Ⅳ）The interlock is valid;

（Ⅴ）Without the movement commands.

（3） MPG select signal

MPG axis select signal (HX, HZ) is conducted with the MPG inserted axis (the contact is closed).

4.3.7.3 MPG Insert Movement

（1） Movement amount

The movement amount of MPG insert is same as the manual hand wheel feeding one. The movement amount depends on the MPG scale and MPG feedrate override（X1, X10, X100）and is added with the automatic running movement amount.

（2） Movement speed

The axial speed during the MPG interruption is the automatic running speed is added with the movement one during the MPG interruption. Therefore, when the axial movement speed exceeds the

rapid traverse rate, the axial one is limited in the rapid movement speed (set by parameters #92（RPDFX）and #93（RPDFX）). Because the part exceeding the rapid traverse rate, the displacement amount and the MPG scale display value become invalid.

（3） The relation between MPG interruption movement and various signals is shown as the following list:

Signal	Movement
Machine lock	Affected. The tool doesn't move when the machine lock is ON.
Display lock	Affected. The relative coordinate values will not change when the display is locked.
X axis mirror image	Not affected. The hand wheel disc is rotated in CW direction, the machine moves in CW.

（4） The relation between the MPG insert movement and the various position display is shown as below:

Display	Movement
Absolute coordinate value	Not affected, the inserted pulse is not added.
Relative coordinate value	Affected, the inserted pulse is added.
Machine position value	Affected, the inserted pulse is added.

（5） Display of movement amount

The movement amount of MPG interruption is displayed on diagnosis (DGNOS) interface.

Sequence number

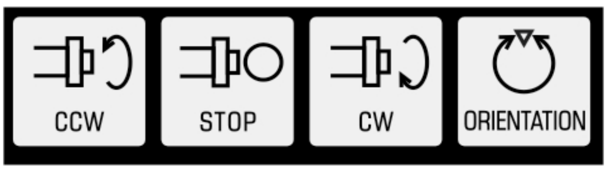
8	0	5	Interruption movement amount of MPG in X coordinate
8	0	6	Interruption movement amount of MPG in Z coordinate

Unit: 0.001mm (mm input)

0.0001inch (inch input)

Note: The movement amount can be cleared with the clear operation.

4.3.8 Manual Spindle Function



In Manual mode, press CCW, the spindle is CCW rotated.

In Manual mode, press Spindle stop, the spindle decelerates and stops.

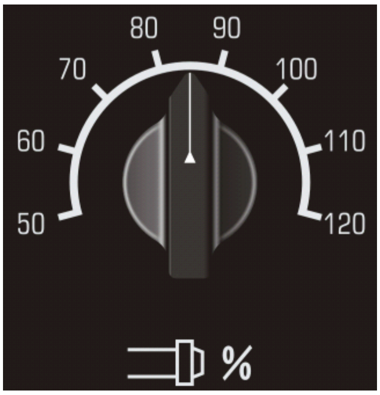
In Manual mode, press CW, the spindle is CW rotated.

Note: The spindle actual revolving direction may vary based on the different machines, please refer to the

machine user manual.

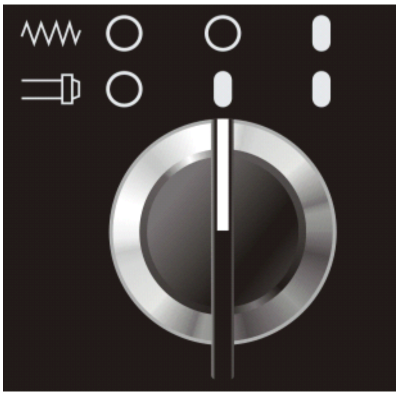
Spindle positioning: When the machine is configurated with the servo spindle drive, press spindle exact stop key in manual mode, or command M19 in Auto, MDI mode, the system will output the positioning start signal into the servo drive. After the drive completes the spindle positioning, it sends the finish signal, and CNC receives the positioning finish signal, the spindle positioning ends. The spindle positioning function is mainly used in tool change and hole boring.

The spindle speed override:



The switch can be adjusted the spindle speed override from 50%～120%.

4.3.9　Spindle Feeding Axis Interlock Switch



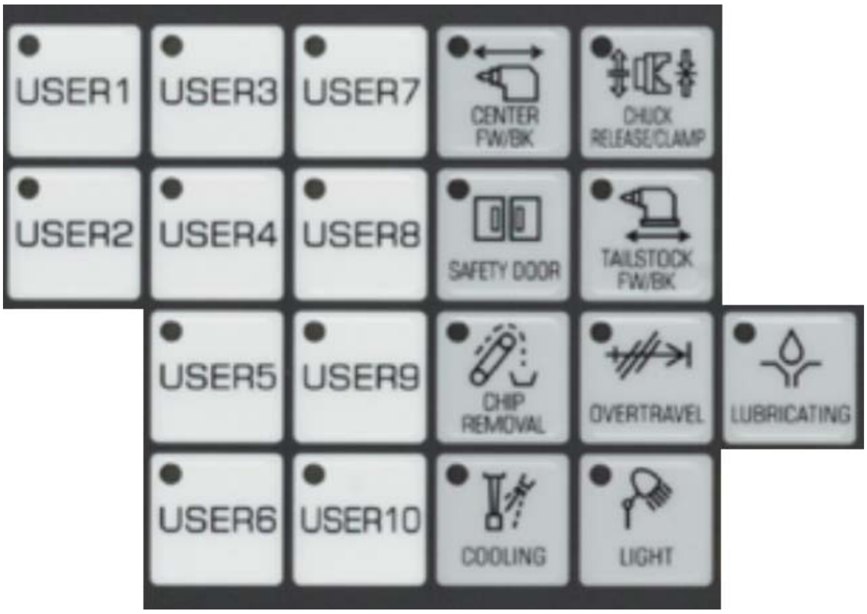
No matter in whatever mode the position marked 0 is OFF and the position marked 1 is ON:

When the feeding and the spindle are ON, the feeding axis and the spindle interlock are invalid;

When the feeding is OFF but the spindle is ON, the feeding axis interlock is valid, while the spindle interlock is invalid, and the screen external information is displayed #2001;

When the feeding axis and the spindle are OFF, the feeding axis and the spindle interlock is valid, and the screen external information is displayed #2002.

4.3.10　Manual Miscellaneous Function



The function and the execution logic of these keys are set by PLC programming, please refer to the PLC user manual of the corresponding version. They respectively control ON/OFF of each part of the machine, pressing it once is ON, and the switch is OFF. USER1～USER10 are the undefined keys, and redefined based on the user requirements.

The tool change key can be used for manual tool change debugging. About the detailed method, please refer to PLC user manual of corresponding version.

And the Nixie tube displays the current tool number.

4.4　GSK983Ta Main Unit

GSK983Ta main unit is shown as the figure below, and includes with LCD screen, MDI keypad and softkey function keypad.

Function buttons

Function buttons display the big item similar with the chapter in the book. Press the function buttons twice, it changes into chapter two; three times, it changes into chapter three (if there are these chapters).

The interfaces in the chapter can be selected by various function buttons, and the interface can be selected by the page keys. The followings are the display and contents of each chapter of each function button.

POSITION	Press once	Display interface of **current position value**
SETTING	Press once	Display interface of **setting**
	Press twice	Display interface of **macro value**
	Press three times	Display interface of **menu switch**
PROGRAM	Press once	Display interface of **program check**
	Press twice	Display interface of **program content**
	Press three times	Display interface of **program**
PARAMETER	Press once	Display interface of **NC parameter**
	Press twice	Display interface of **PC parameter**
OFFSET	Press once	Display interface of **tool wearing offset**

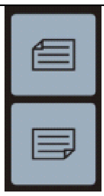
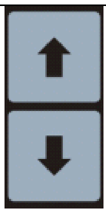
	Press twice	Display interface of **workpiece coordinate offset**
	Press three times	Display interface of **tool outline offset**
	Press once	Display interface of **alarm information**
	Press twice	Display interface of **external alarm information**
	Press once	Display interface of **current block command**
	Press twice	Display interface of **program restart**
	Press once	Display interface of **NC diagnosis information**
	Press twice	Display interface of **tool lifetime management**
	Press once	Display interface of **figure menu**
	Press twice	Display interface of **drawing**
	Chapter one	**Wait for the external data sending internal, the interruption level is the highest, and the other function keys are invalid.**
	Chapter one	**Send the system data to outside, the interruption level is the highest, and the other function keys are invalid.**

Note: Press  with some function button meanwhile to cancel the interface display, and then press the other function button to display the corresponding interface.

Introduction of other keys in MDI keypad

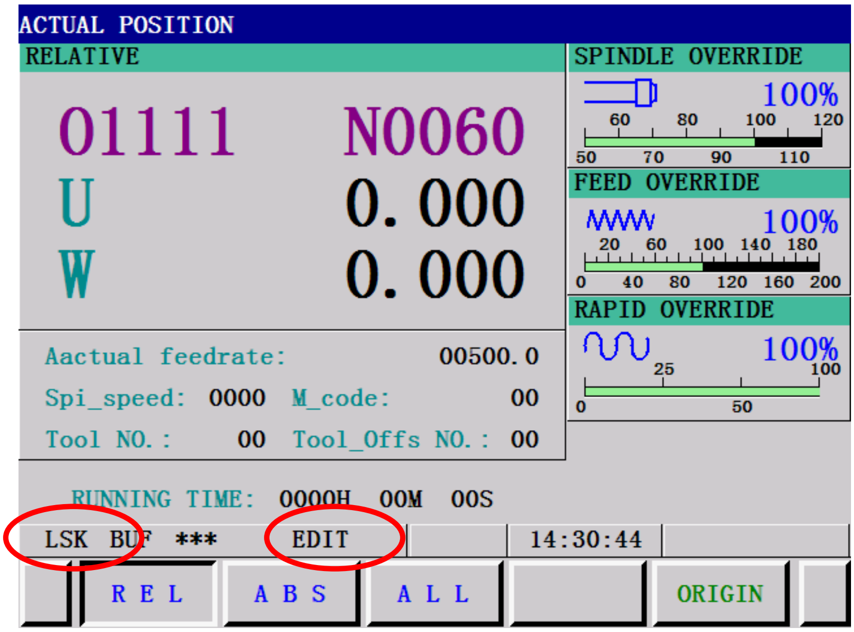
Reset key 	Press this key to reset CNC and clear the alarm.

Input key 	After pressing the address key or the numerical key, the input data just display on the screen and are in the buffer. After pressing this key, these data are read into the memory.
	Press cancel key to cancel the data in the buffer from the last character.
Edit keys 	The three keys can be executed the replace, insert and delete.
	Some address keys include two characters, press shift key to select. When “ ^ ”character is displayed on the screen, the character in the lower-right corner can be input.
Address keys 	Press numerical keys to input the letters.
Numerical keys	Press numerical keys to input the digits and the punctuation marks. Press EOB key to input colon “:”.

	
	Page up/down key: The interface is paged up or down for one page.
	The cursor is moved upward or downward.
	Soft keys: The function of soft keys is to display the content at the bottom of the screen.

4.4.1 Status Display

In the lower-left corner, the status is displayed.



The status are displayed as below:

NOT READY: It means the control device or the servo system is abnormal.

LSK: It means the label skip state. It changes into the label skip state by turning on the power or resetting the system not in MDI mode.

BUF: It means that some block has been read but not executed. The block not executed is cleared by resetting not in MDI mode.

ALM: It means the alarm has occurs, press ALARM to find the alarm content, and it is twinkling.

EDIT: It means the executing is being performed or is to be edited.

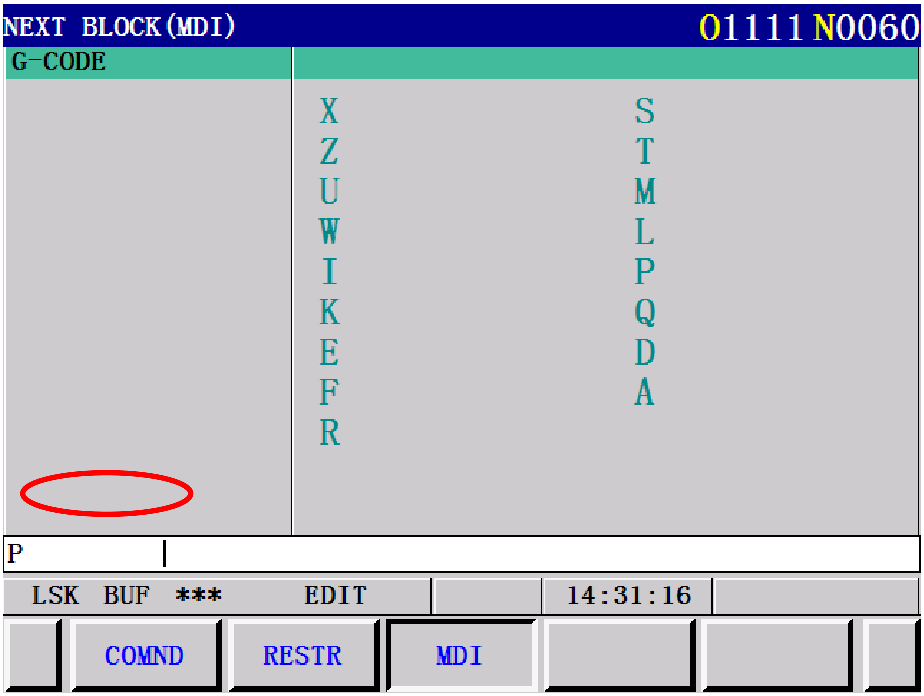
Select the different switch based on the different modes, there is (AUTO) automatic, (MDI)manual data input, (HNDL) hand wheel, (JOG) Jog mode and machine zero return (not displayed).

SRCH: It means the present program number is being searched, and it is twinkling.

RESTR: It means the period from the program restarting to the last axis return, and it is twinkling.

4.4.2 Key Input Display

The content input by the address and the numerical keys are displayed in the left-lower corner of the screen.



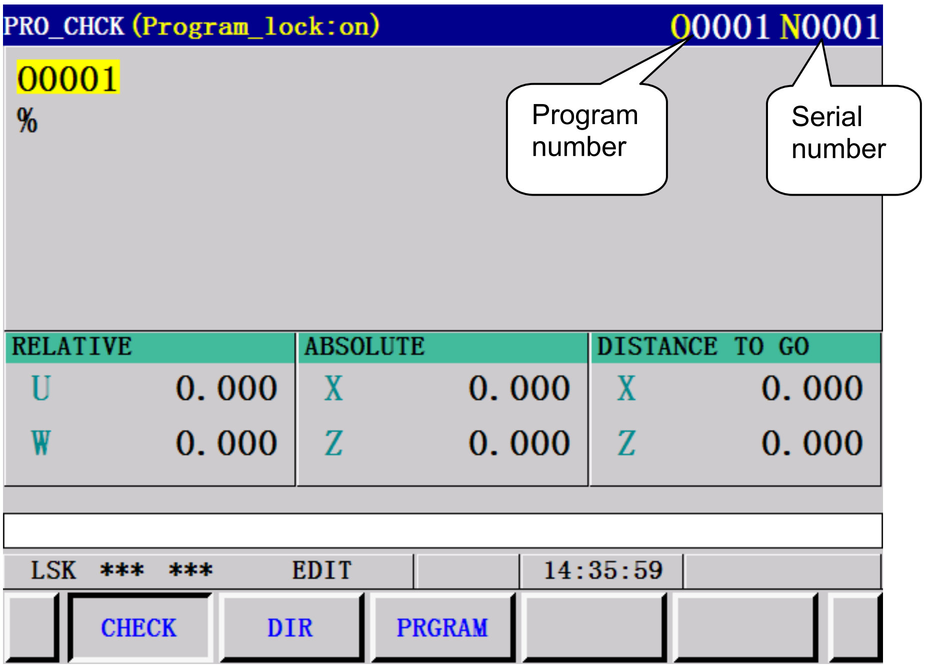
The data input can't be executed in position or alarm screen.

Except for the program editing, one word composed of one address and the digits can be input. Press CANCEL can clear one input word. During the program editing, it is not limited in one word but in 32 words.

The input last word can be cleared by CANCEL once, and the input words can be cleared one by one from the behind by continuously pressing CANCEL.

4.4.3 Display the Program Number and the Sequence Number

The program number and the sequence number are displayed on the top part shown as the following figure.

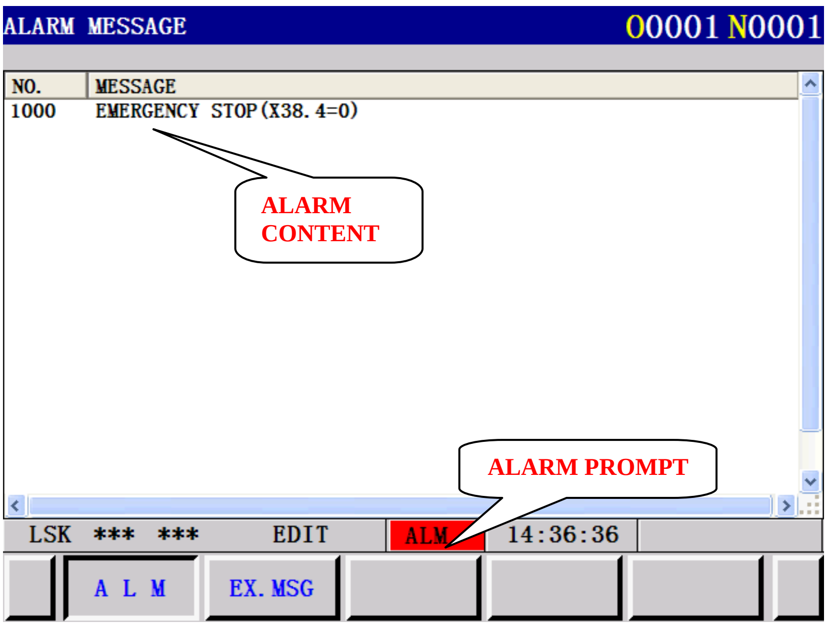


4.4.4 Alarm Display

When the alarm occurs, ALM is displayed on the lower-right corner, and the alarm content can be learnt with the following operations.

Press ALARM , the operation information occurs, press ALARM again, the alarm content is displayed as the figure below.

About the content of the alarm number, refer to Appendix 7.



Note: When the alarm occurs, the alarm content is automatic displayed on the screen.

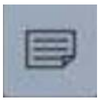
4.4.5 Operation Information

Mode	Operation	Content
Except for the Edit mode	Except for the following situations	Display the last executed sequence number
	Searching the sequence number	Display the read sequence number at random during searching
In Auto mode	Press ↑ when the function button is in program state .	Return to the head of the current program and display the program number (it can't return during the program running).
Edit mode	Continuously press ↓ when the function button is in program state .	Look the position from the current position of the memory in CW direction, and the system displays the initial N value.
	Press ↑ when the function button is in program state .	Look the position from the current position of the memory in CCW direction, and the system displays the initial N value.
	Press reset when it is in resetting state .	Return to the head of the current program, and display the current number.
In Auto mode	Searching the program number	Display the searched program number.

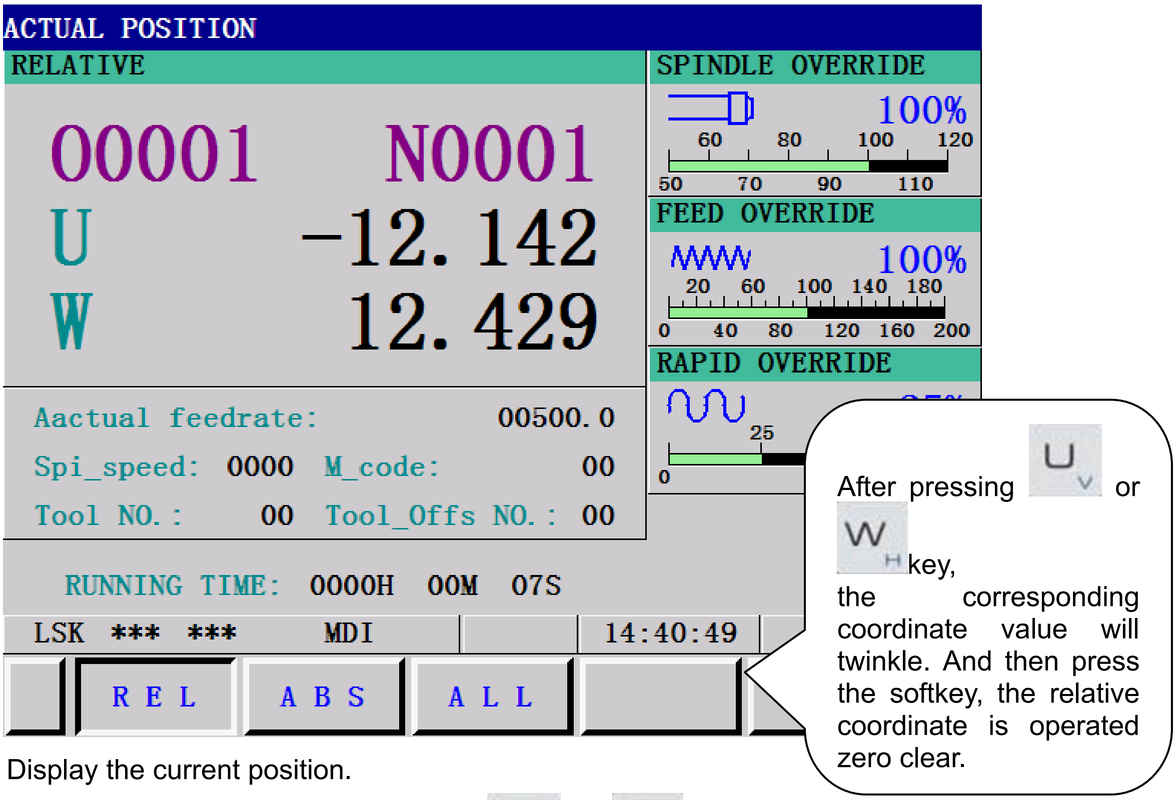
4.4.6 Current Position Display and Resetting

（1） Press the position button 

;

（2） Press page button  , the three situations are displayed.

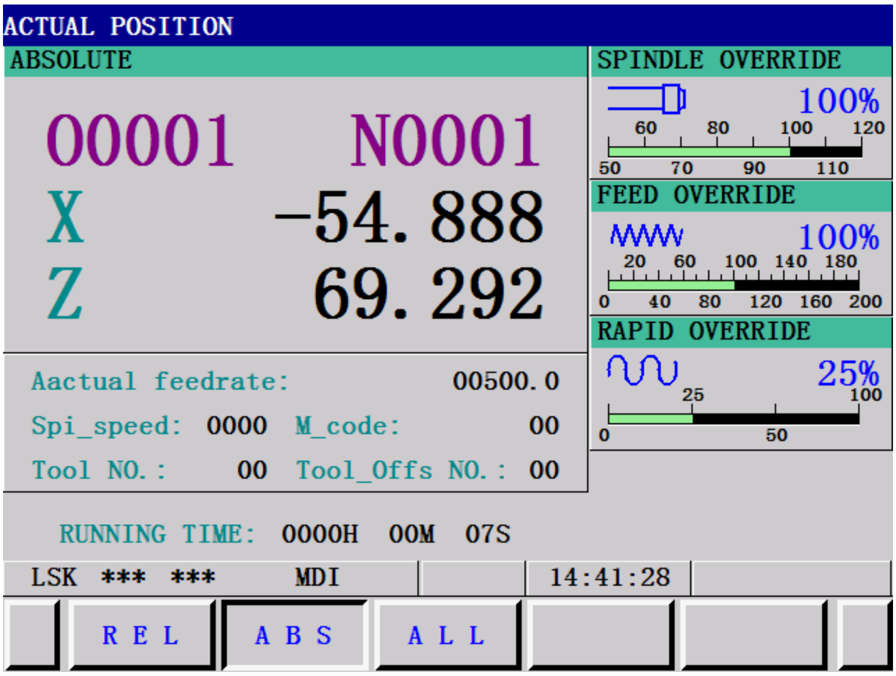
（Ⅰ） Position display of the current coordinate system



Display the current position.

Relative coordinate zero clear: Press U or W key when zero clear is displayed in the state. Then, the pressed address display is twinkling. And press SHIFT, then, the relative position of the twinkling address is cleared as zero.

（II）Position display of the workpiece coordinate system



Automatic coordinate setting G50 programming coordinate system is displayed or the present value of the coordinate system is set by resetting shown as below.

（III）Display the total position

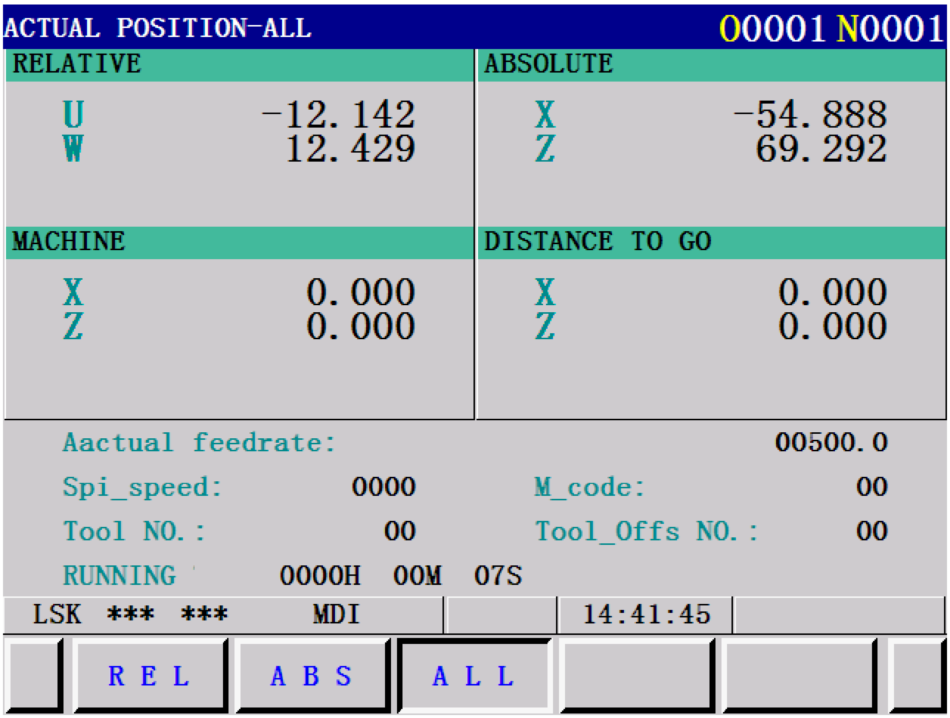
The current position of the following coordinate can be displayed meanwhile.

（a）Position of the relative coordinate system（RELATIVE）

（b）Position of the absolute coordinate system（ABSOLUTE）

（c）Position of the machine coordinate system（MACHINE）

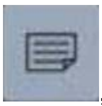
（d）Remaining distance（DISTANCE TO GO）



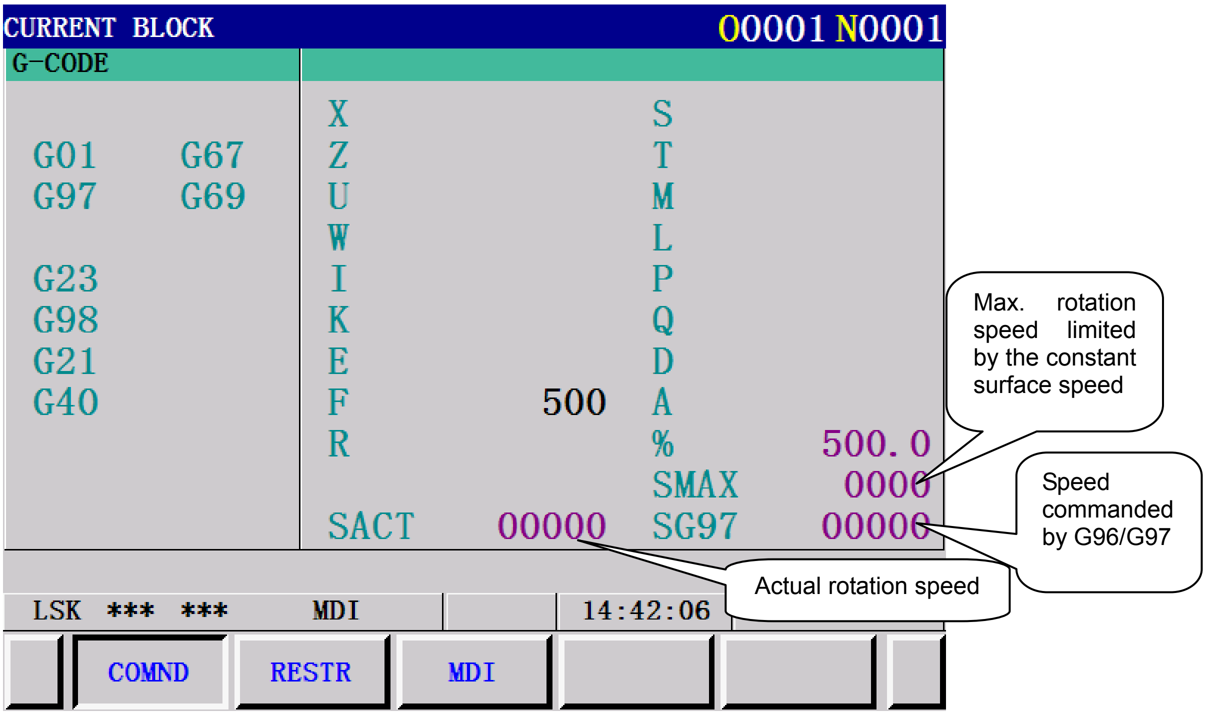
Distance to go is the commanded movement amount not executed in one block.

4.4.7 Display the Commanded Value

（1） Press the command button .

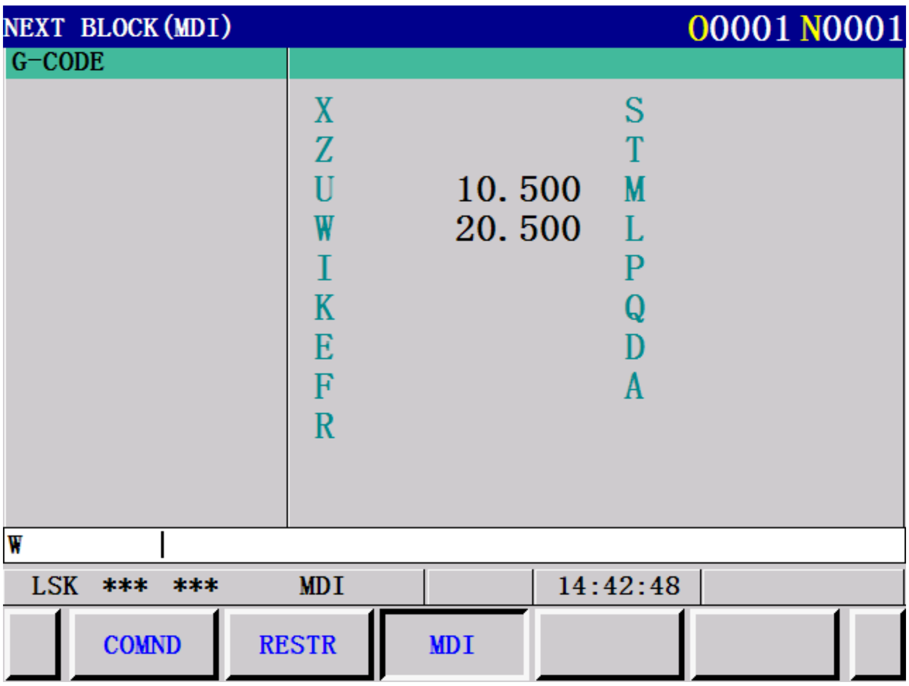
（2） Press the page button , the following three situations are displayed.

（Ⅰ）The executing commanded values and the previously commanded modal values are displayed.

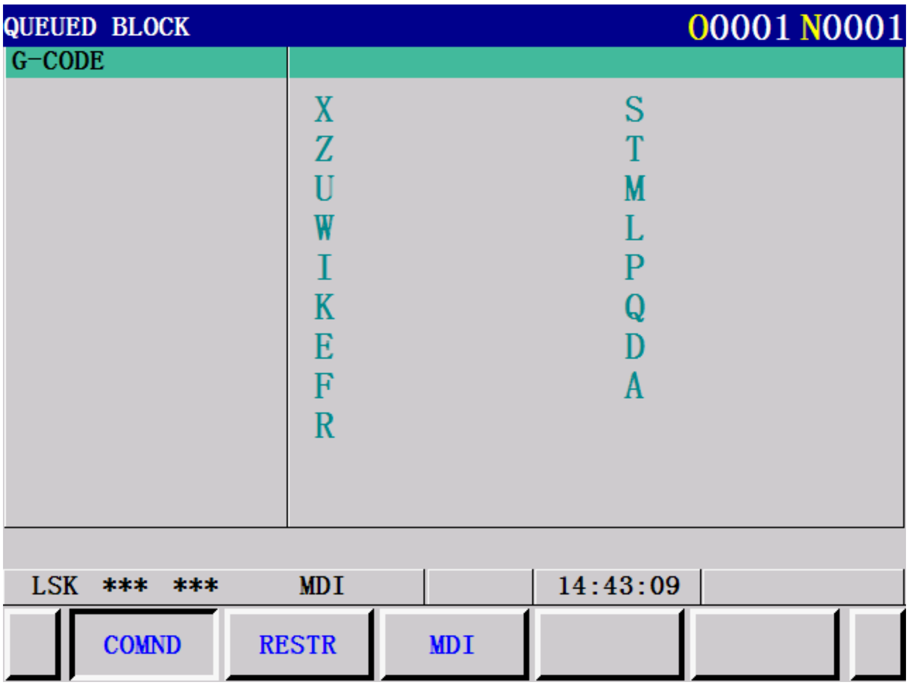


Note: The numerical values after % is the feedrate which the feedrate multiplied by the override.

（II）Display the commanded value input from MDI, or the commanded value to be executed.



（III）Display the next commanded value to be executed in the tool nose R compensation.



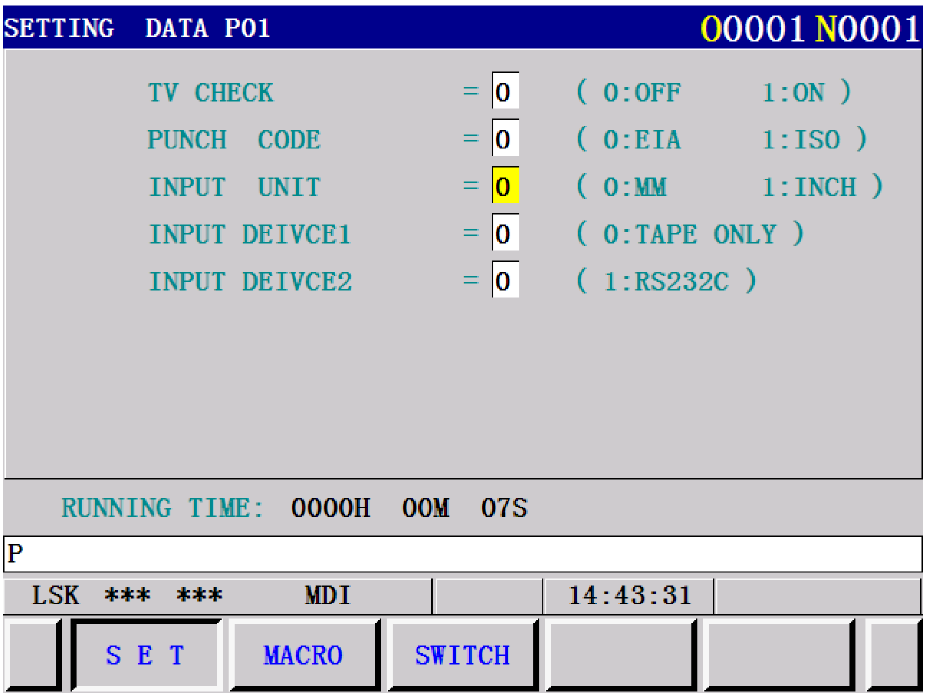
4.4.8　Setting (Function Setting)

4.4.8.1　Input, Output, Other Display and Setting

（1）　Press setting button .

（2）　Press page button , the following two situations are displayed and set.

（Ⅰ）Setting and display the input and output

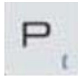


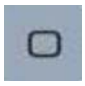
Setting (the program protection lock can’t be executed if it is OFF; or it can be executed even it is

OFF and the two status can be switched with parameters.)

（a）The mode selection is in MDI mode

（b）Press  or , the cursor is moved to the item to be changed (the cursor can't be moved with address N).

(c) Based on the following list, input 1 or 0. Press the addresses 



or 

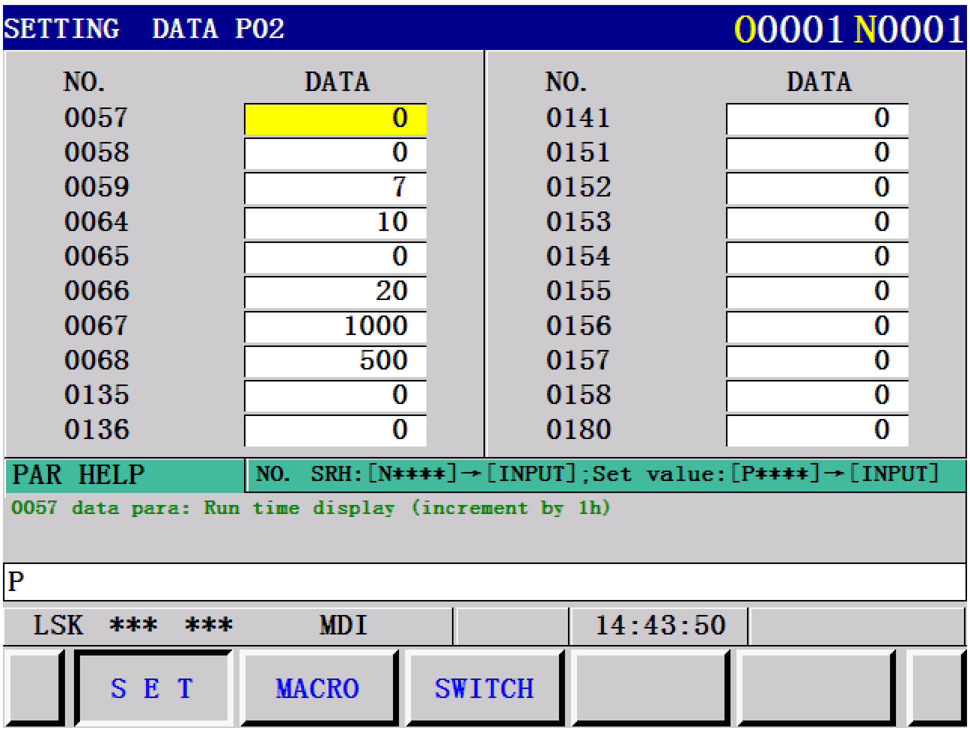
to input.

Display / Item	0	1
TV CHECK	TV check is not performed.	TV check is performed.
PUNCH CKDE	EIA code is output.	ISO code is output.
INPUT UNIT	MM input	Inch input
INPUT DEVICE1	The standard is set as 0.	
INPUT DEVICE2	The standard is set as 1.	

Note 1: The selection function not chosen can't be used for setting. For example, without the selection of the conversion between inch/metric system, in the metric machine, INCH=1 can't be set; without ISO code input selection, ISO=1 can't be set.

Note 2: After executing G20 (inch input)/G21 (metric input) command, the above content of INPUT UNIT is automatically changed.

（II）Other setting and display



The displayed numbers and the set content is as below:

Data number	Content
057 *	Running time (based on the hour/unit) （TMHOR）

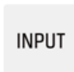
058 *	Revolving time (min/unit) （TMMIN）
059 *	Running time (sec/unit) （TMSEC）
064 *	Chamfering width of thread（THDCH）
065 *	The return amount of G74,G75（GROVE）
066 *	The finishing surplus of G76（THDFN）
067 *	The retraction amount of G71，G72（MRCDT）
068 *	The least remaining amount of G76（THCLM）
141 *	Running time （TIME）
151 *	The X coordinate value of the 1st peak in the stored stroke limit 2
152 *	The Z coordinate value of the 1st peak in the stored stroke limit 2
153 *	The X coordinate value of the 2nd peak in the stored stroke limit 2
154 *	The Z coordinate value of the 2nd peak in the stored stroke limit 2
155 *	The X coordinate value of the 1st peak in the stored stroke limit 3
156 *	The Z coordinate value of the 1st peak in the stored stroke limit 3
157 *	The X coordinate value of the 2nd peak in the stored stroke limit 3
158 *	The Z coordinate value of the 2nd peak in the stored stroke limit 3
180 *	Sequence number comparison and stop
319 *	Various setting（PRG8，MSBL）
340 *	Reserved by the manufacturer
341 *	Reserved by the manufacturer

Note 1: The data series numbers except for the ones in the list are not displayed.
Note 2: Setting the parameter number and meaning same with the NC parameter number and its meaning.
Note 3: About the relative content remark, please refer to the introduction of parameters in Appendix 5.

Setting (the program protection lock can't be executed if it is OFF; or it can be executed even it is OFF and the two status can be switched with parameters.)

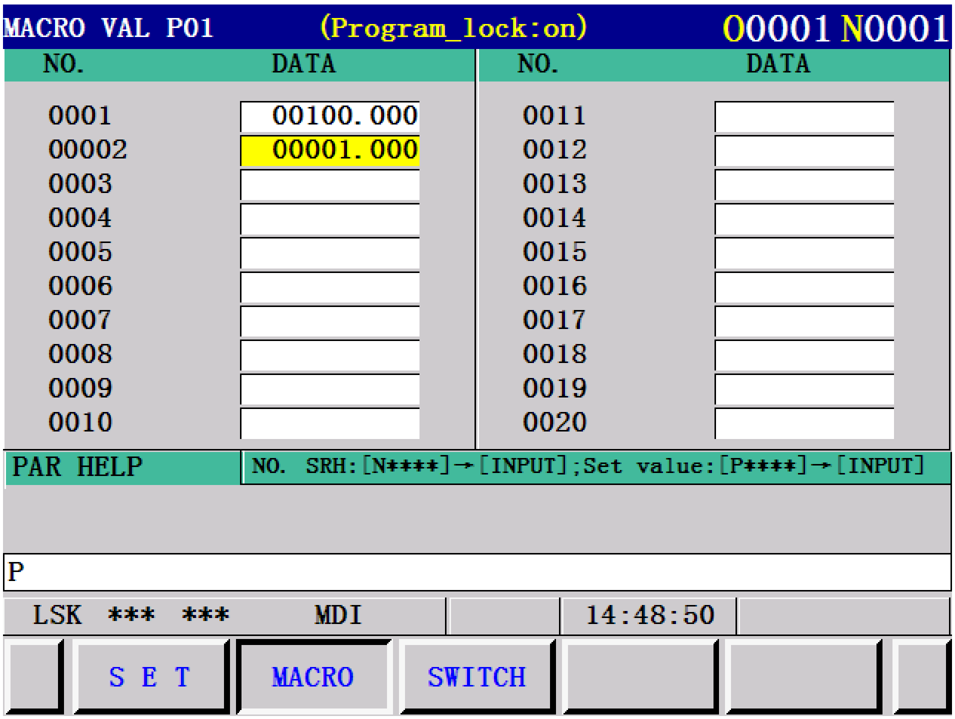
（a）Mode selection is set in MDI mode

（b）Press  or  to move the cursor to the item to be changed (the address N can't be moved by the cursor.)

（c）Press the address  the numerical value to be set , and then press  to be input.

4.4.8.2 Display and Setting the Custom Macro Variable Values

All of the common variables and the local variable values of the macro itself which is called currently can display on LCD.



The macro variable values (void) (refer to <undefined variable> of 10.2.3 of chapter three) is displayed as blank.

When the absolute value exceeds 99999999, the system displays ±OVER FLOW.

When the absolute value is not 0 but be less than ±0.0000001, the system displays ±UNDR FLOW.

Display

（1）Select the set chapter two (the method is: press [set] to display the setting interface, and then press [set] once more.

（2）There are six interfaces, press [page] button to display the required interface.

Page 1: Local variables #1～#20.

Page 2: Local variables #21～#33.

Page 3: Common variables #100～#119.

Page 4: Common variables #120～#139.

Page 5: Common variables #140～#149.

Page 6: Common variables #500～#509.

（3）Search for the macro variable number position.

Method 1: Press [↑] , [↓] button to move the cursor in turn. When the cursor exceeds the interface the system turns over into the next one.

Method 2: Press [N] [macro variable] [input] .

（4）Setting the macro variable value

（a） Select MDI mode.

（b） Display the variable to be changed, and move the cursor into the variable number to be changed, and then input [P] [macro variable value] [input] .

4.4.9　MDI Operation

Input and execute the commands of one block from MDI panel. MDI operation is for the simple debugging operation.

MDI operation steps:

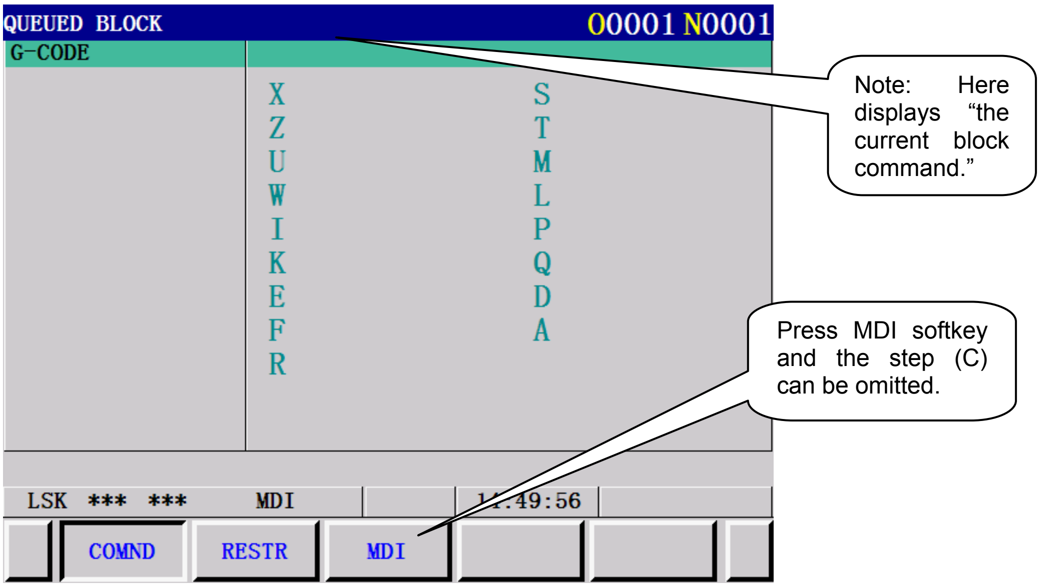
(1) Take the example of running one block U10.5　W 20.5.

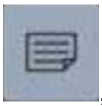
（a）Press MDI button 

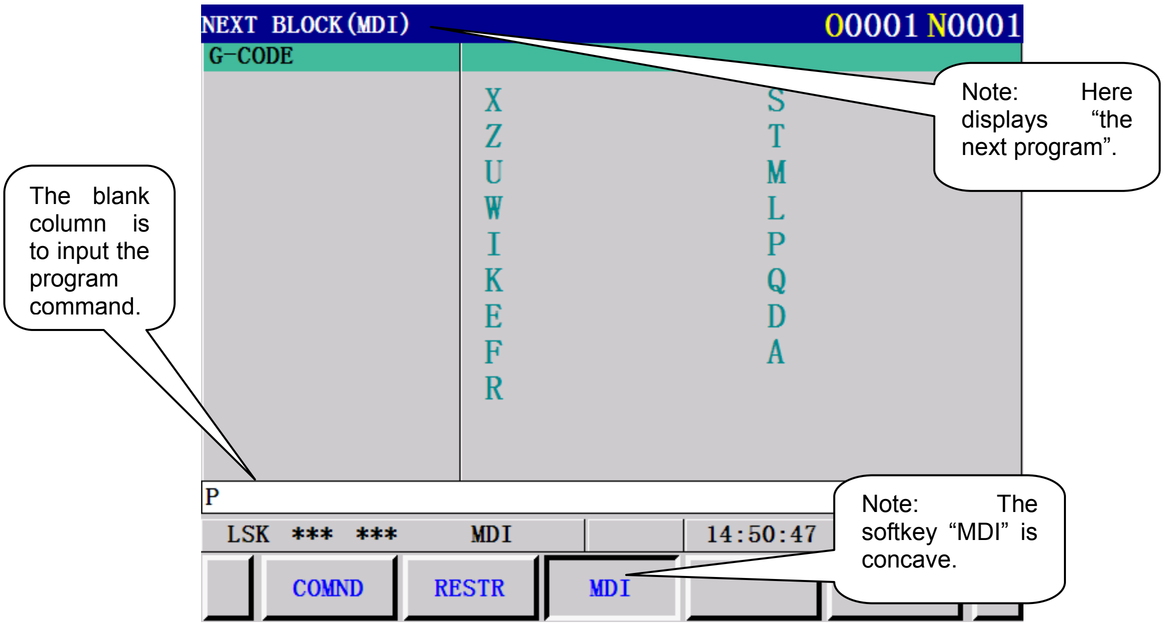
on the operation panel.

（b）Select the function key to choose the command button 

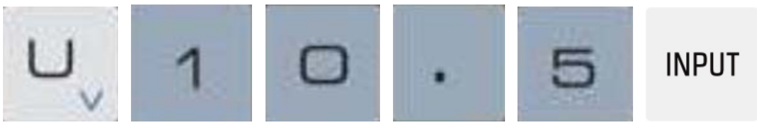
, the system will pop up the following interface:



（c） Press the page button , the interface of “the next block” will pop up, which is shown as the following figure:

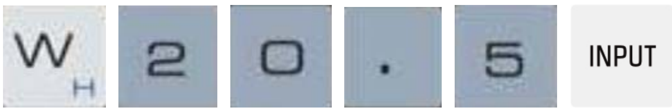


(d) Press the following keys in turn on the panel.

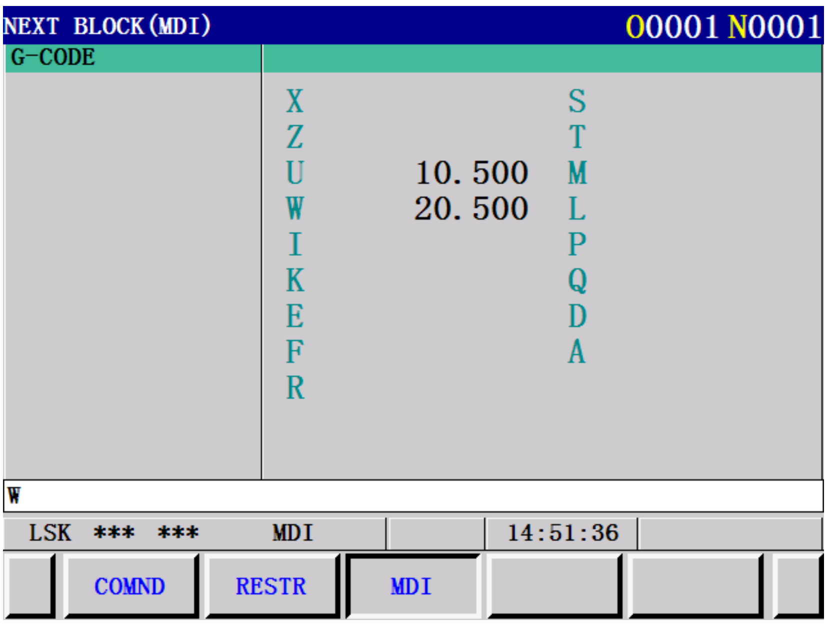


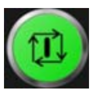
U10.5 is input.

(e) Press the following keys in turn on the panel.



W20.5 is input, which is shown as the following figure:



(f) Press the cycle start button  on the machine operation panel to run the block.

（2）Before pressing cycle start, clear Z 200.5 in X 10.5 Z 200. That is to say, only run X10.5.

（a） Press based on the following sequence: Z cancel input.

（b） Press cycle start

（3）Clear the modal data.

Because G code mode, and F, E and T can't be cleared, so the right modal data should be input, again to correct.

4.4.10　MDI Operation Start

Press cycle start , execute the command input by MDI.

4.4.11　Resetting

Reset button 

Press the reset button 

. Normally, it is used for clearing the alarm state.

After pressing the reset button 

, NC is changed into the following set.

State before resetting		State after resetting
During executing the movement commands		Deceleration stops and the remaining movement is cleared.
M, S and T output		The output sequence stops, and please refer to the machine user manual for the machine side.
One block has read into the buffer register	MDI mode	The content of the buffer register can't be cleared.
	Modes except for the above mentioned	Clear the content of the buffer register and the display of BUF is cleared.

No matter in any situation, press the reset button 

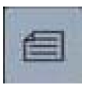
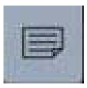
, NC system is set as resetting state. In Auto mode, it is changed into the skip state.

4.4.12　Setting and Display the Tool Position Offset Amount, the Tool Nose R Compensation Amount

4.4.12.1　Input the Absolute Value

（1）　Press 

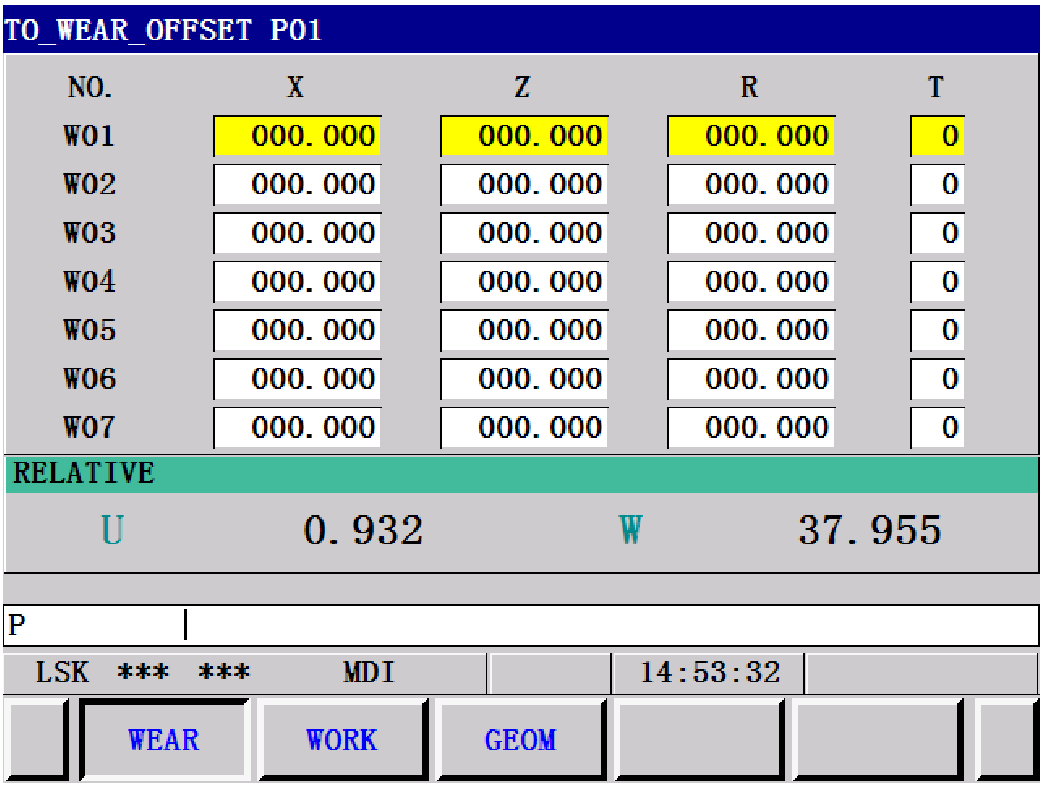
key.

（2）　Because the system displays many pages, press  or  to select the required page.

Page 1　　Offset amount　　1～7

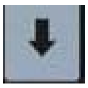
Page 2　　……　　8～9 or 8～14（Select）

Page 3 ...... 15～21（Select）
Page 4 ...... 22～28（Select）
Page 5 ...... 29～32（Select）



Display of the offset amount on the 1st page

（3） Searching the offset number: Move the cursor to the position of the offset number to be rewritten.

Method 1: Press  and  keys, the cursor is moved in order, and it is moved into the next page if it exceeds one page.

Method 2: After inputting N , offset number, press INPUT.

（4） Select in any mode.

（5） Input the offset amount:

Input: X, Z, R or T + offset amount, press INPUT key.

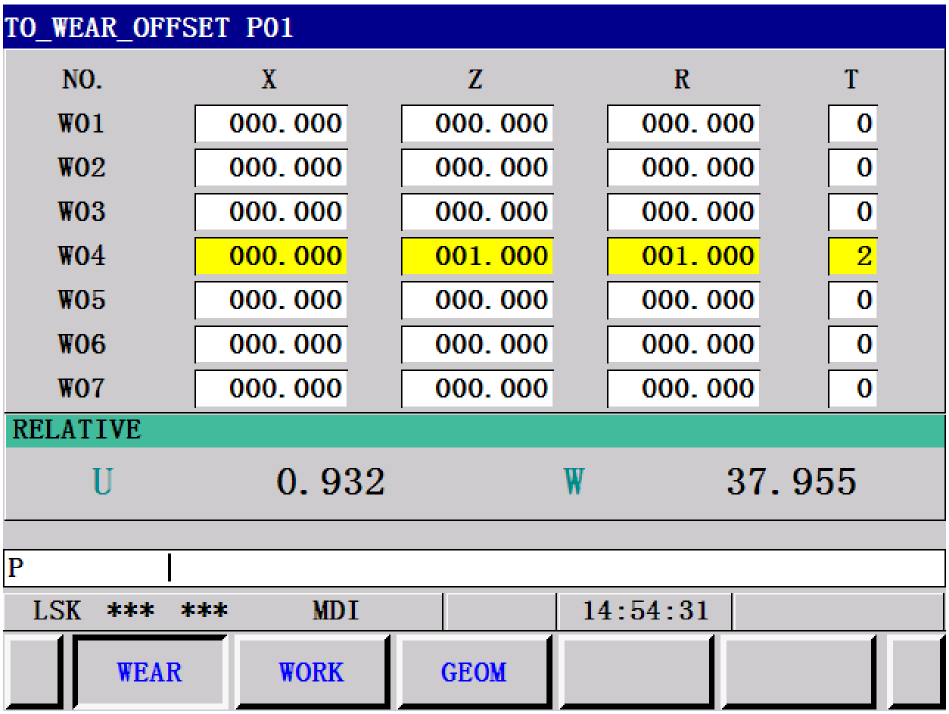
X or Z+ offset amount, press INPUT key: Input the absolute value of X axis or Z axis offset amount;

R+ offset amount, press INPUT key: Input the absolute value of the tool nose R offset amount;

T + offset amount (tool number) press INPUT key: the assumed tool nose number.

Example: Input at the offset number #W04: display Z 1 . 0 , R 1 . 0 , T 2 input , which is shown as the following figure.

Chapte 4 Operation



Note 1: The value of the tool wearing offset is limited by NC parameters #388 and #358.
Note 2: Input only when the letters on the lower-right corner don't twinkle; otherwise, input NO, and then, the letters don't twinkle.
Note 3: T is represented as the tool number, R as the radius.

4.4.12.2 The Incremental Value Input

Through the selection function, the offset of the incremental value can be input, then, the increment can be input with [U] [W] .

[X]
[Z]
[U]
[W]
[R]
[T]
} [Offset number] [Input]

[X] , [Z] : The offset amount of X and Z axes (input by the absolute value).

[U] , [W] : The offset amount of X and Z axes (input by the incremental value).

[R] : Tool nose R offset amount (absolute value input/incremental value input).

[T] : The assumed tool nose number

Whether the input value of R is absolute or the incremental is set by NC parameter #7.6（IOF）.

Note: When NC parameter #7.6 is 1, R value is the incremental input.

The incremental offset input is the added or subtracted part of the input offset.

（1）The incremental offset input

When the current tool nose R offset amount is 5.678, input [R] [1] [.] [5] [input], the offset

amount is changed into 7.178.

（2）The absolute value should be input when the incremental offset is input:

The current offset amount is 5.678, press [R] [cancel] [input] , R offset amount becomes 0. Then, after pressing [R] [1] [.] [5] [input], R offset amount is changed into 1.5.

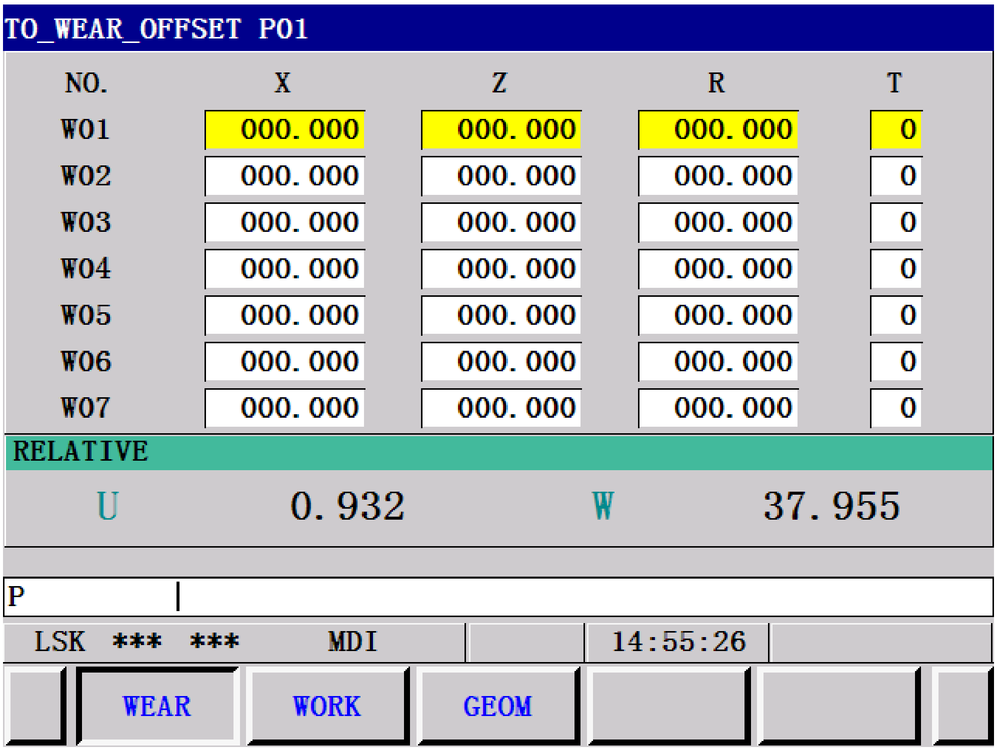
Note 1: During the automatic running, the offset amount is changed, the new offset amount becomes valid from the next block or from the next specified T code is set by parameter （#8.4 TLCC）. (When NC parameter #8.4 is 1, the new offset becomes valid from the next program; 0, valid from the next T code.)

Note 2: Input only when the letters on the lower-right corner don't twinkle; otherwise, input NO, and then, the letters don't twinkle.

4.4.12.3 Respectively Setting Tool Figure Offset and Tool Wearing Offset

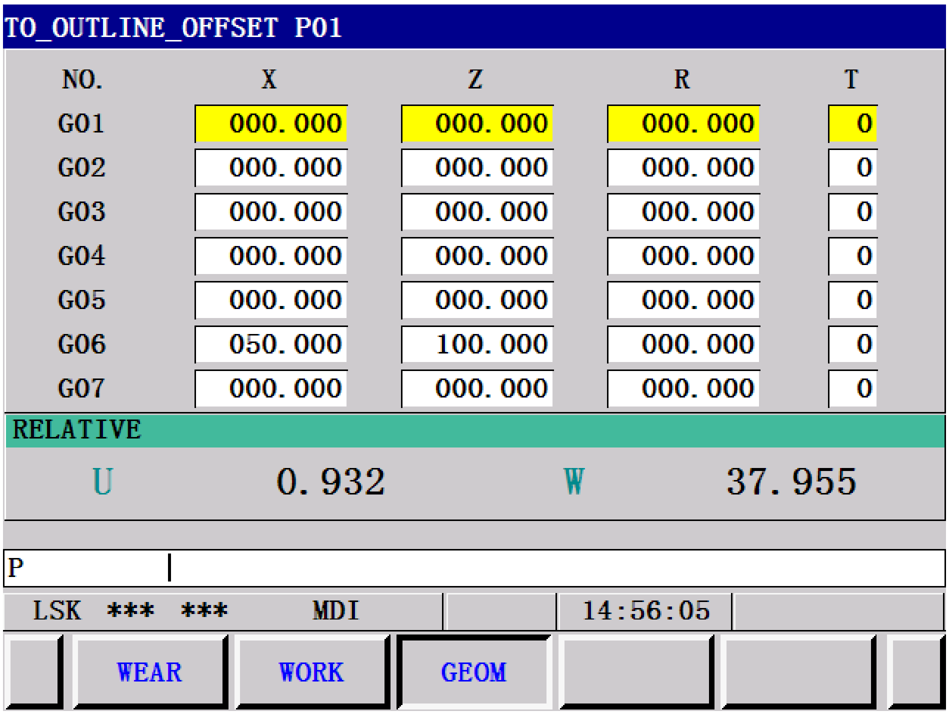
（1） Display the tool wearing offset

On OFFSET interface, the 1st chapter is displayed (press OFFSET key for several times or the softkey WEAR, the following interface occurs.).



（2） Display the tool outline offset:

On OFFSET interface, the 3rd chapter is displayed (press OFFSET key for several times or the softkey GEOM, the following interface occurs.).



Tool outline offset figure

Setting

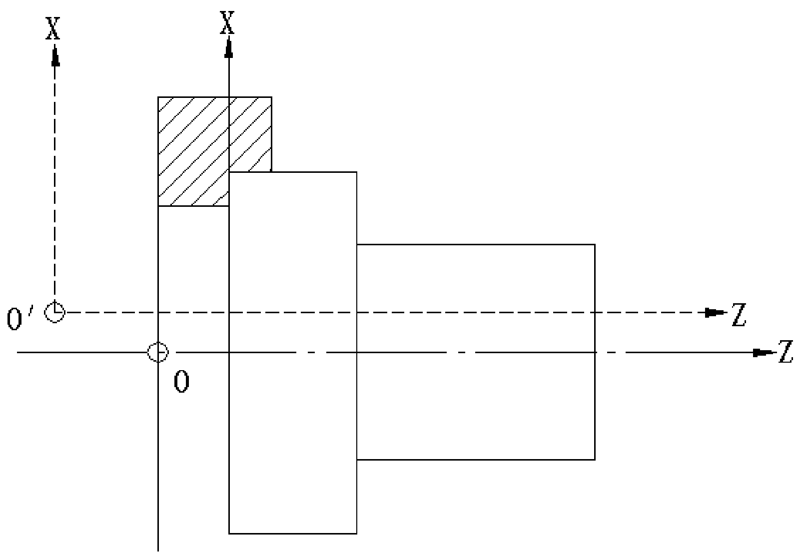
（Ⅰ）Display the corresponding interface by pressing GEOM or WEAR.

（Ⅱ）The setting is same with that of **5.12.1** and **5.12.2**.

4.4.12.4 Setting the Workpiece Coordinate Offset

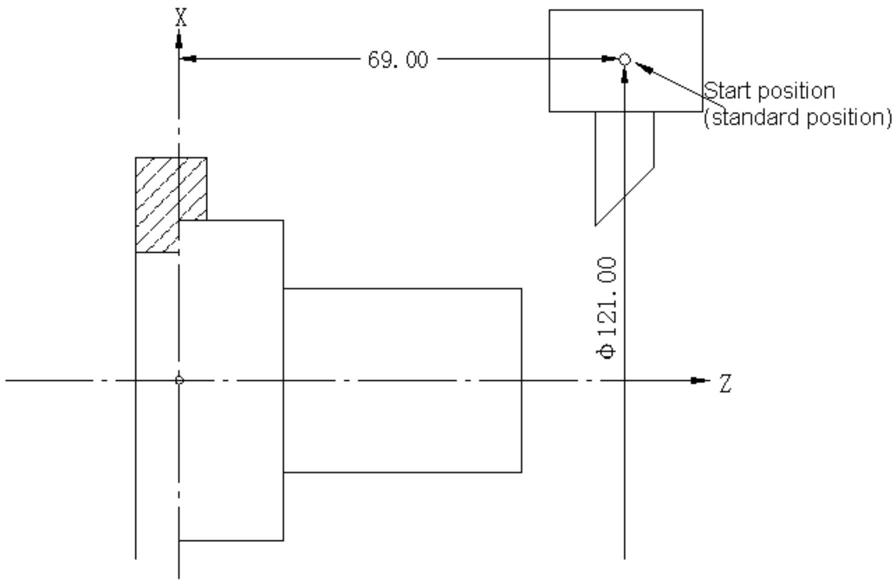
The coordinate offset is set when the difference exists between the workpiece coordinate system during programming and the ones commanded by G50 and set by the automatic coordinate system.

The offset amount is set in the offset number 00 of the workpiece coordinate, and the setting method is same with setting the tool position offset amount.



Offset amount from 0' to 0
is set in the offset number 00

XZ – 0 Coordinate system during programming
XZ – 0′Coordinate system is set currently
（Offset the coordinate system which can be corrected）

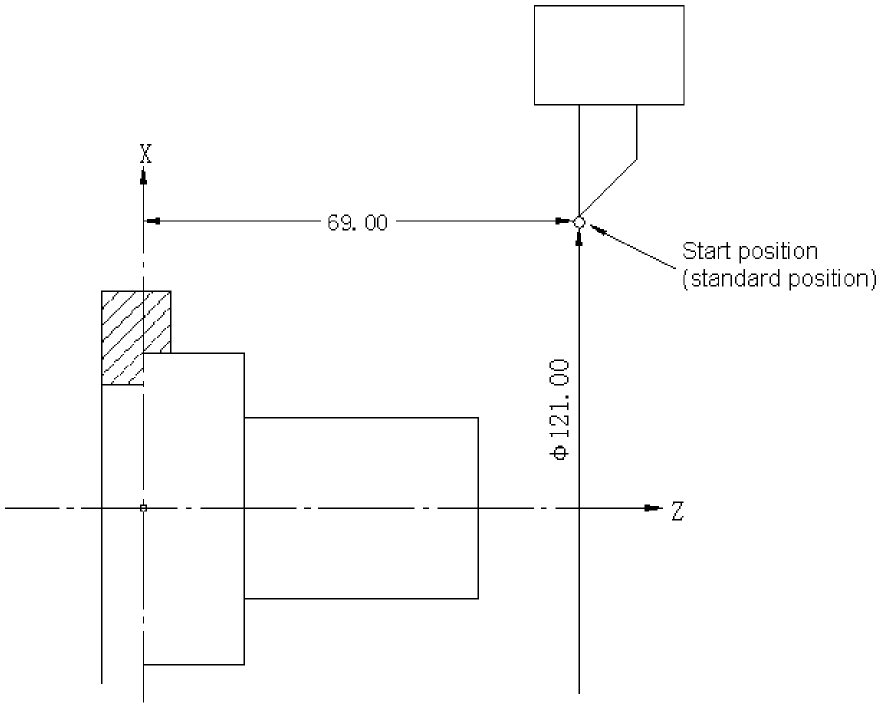


G50 X120.0 Z70.0; (diameter command)

Expect the distance of the standard point position off the workpiece origin is X=120.0（diameter value），Z=70.0（offset amount is 0）which is shown as the above figure, but the actual one is X=121.0（diameter value），Z=69.0, please input the workpiece coordinate offset amount;

X=1.0 Z=－1.0

Then, the expected coordinate system can be obtained.



G50（G92）X120.0 Z70.0; (diameter command)

In the above figure, the standard point is on the top of the standard tool, if the distance of the standard point position from the workpiece origin is X=120.0（diameter value）Z=70.0, but the actual

one is X=121.0（diameter value）Z=69.0, the workpiece coordinate offset amount is input:

X=1.0 Z=－1.0

Then, the expected coordinate system can be obtained.

Setting the actual offset amount uses the method of the workpiece coordinate system offset direct input in 5.12.5, which is very simple.

Note 1: The workpiece coordinate offset becomes valid immediately after setting the offset amount.

Note 2: After setting the workpiece coordinate offset amount, the coordinate system is set by G50, the workpiece coordinate offset amount doesn't function.

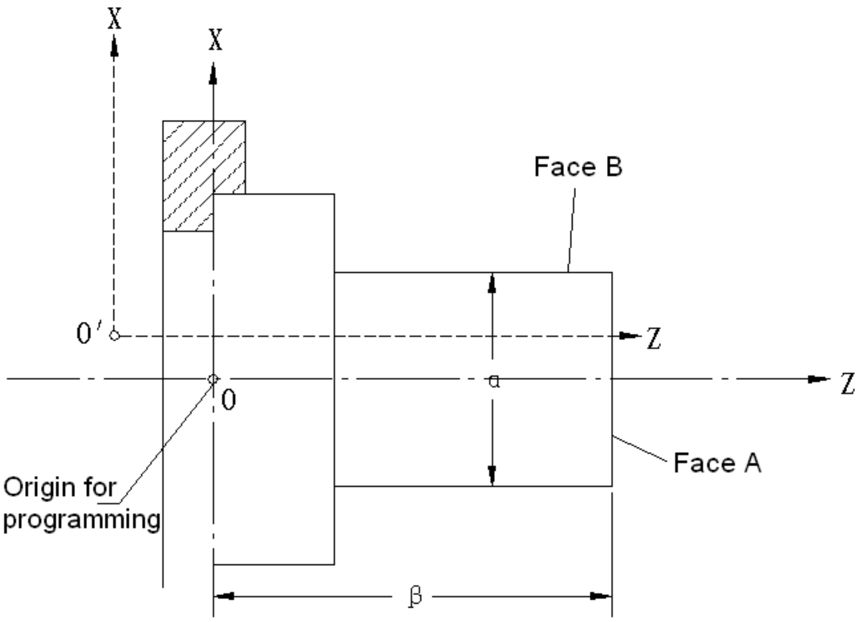
Example: G50 X=100.0 Z=80.0 has been set, no matter how much the offset amount is, the present tool standard position X=100.0 Z=80.0 is set as a new one.

Note 3: After setting the workpiece coordinate offset amount, the manual reference position return is executed, if there is the function of the automatic coordinate setting, it functions and the set coordinate system is offseted immediately.

Note 4: The offset amount of X axis is the diameter value or the radius one is specified by the diameter or the radius of the part program.

4.4.12.5 Direct Input the Measured Value of the Workpiece Coordinate System

As the following figure shows, the coordinate system can be offseted with the direct measuring value when there is difference between the coordinate system used in programming and the one commanded by G50 and set by the automatic coordinate system.



（a）Cut face A in manual mode with the standard tool.

（b）After pressing 

on the machine operation panel, the tool leaves face A, and the spindle is stopped revolving.

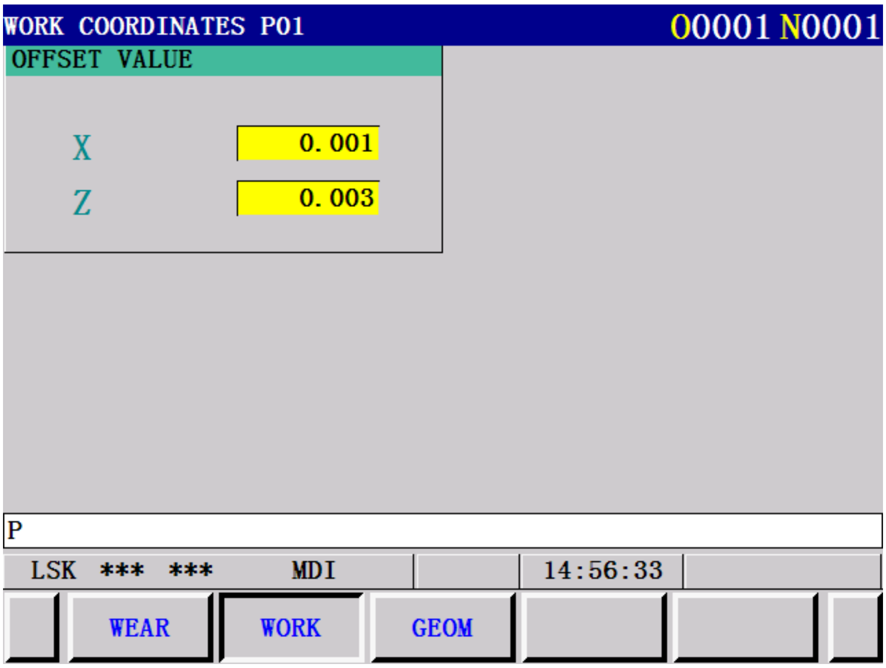
（c）Measure the distance β from the programming origin to face A.

（d）Press 

. If the interface of the workpiece coordinate offset is not selected, press 

once more.

（e）Input N 1 0 0 input，the input N is twinkling.



Workpiece offset interface

(f) Input [Z] [measuring value β] [input], and then, the offset amount is input into the place of Z of the workpiece coordinate offset.

(g) Cut face B in Manual mode.

(h) After pressing 

button on the machine operation panel, the tool leaves face B and the spindle is stopped.

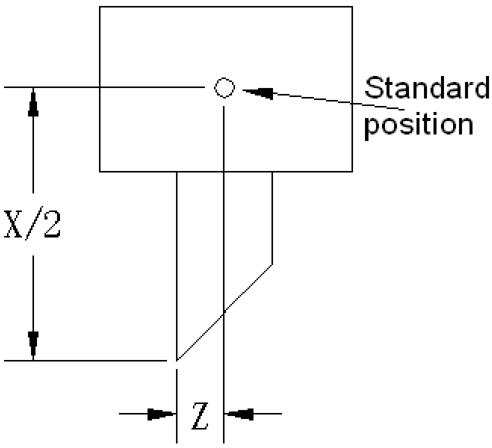
(i) Measure the diameter α on face B.

(j) Input [X] [measuring value α] [input], and then, the offset amount is input into the place of X of the workpiece coordinate offset.

Through the above operation, the offset amount of 0′→ 0 is automatically set in the workpiece coordinate offset, meanwhile, the actual set coordinate system complies with the programmed one.

Then, the offset amount of the standard tool is 0, that is to say, the tool nose is taken as the origin of the workpiece coordinate system, then, the coordinate system of X=0，Z=0 is set.

Note 1:



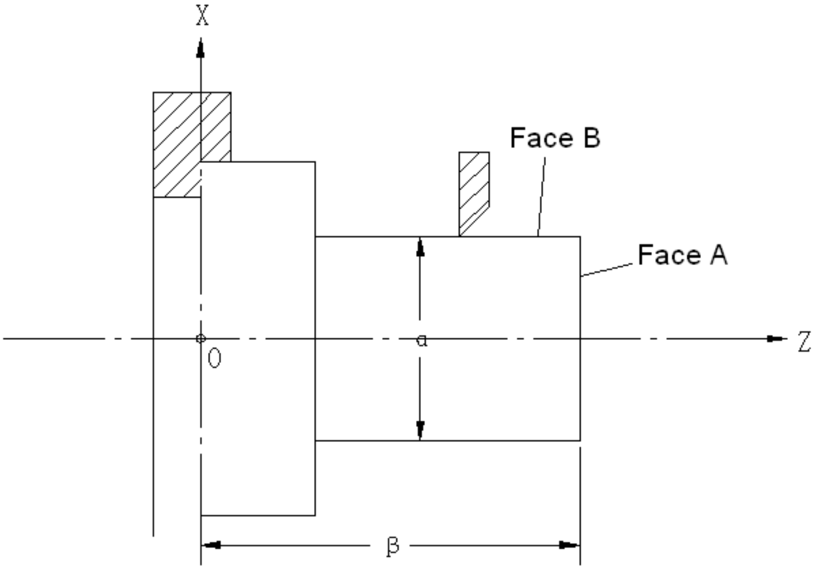
When the standard tool is set with the offset amount of X, Z in the above figure and the offset is valid (TXXXX is commanded), before cutting the previously mentioned faces A and B, the setting α, β is completed. And the coordinate system X=0, Z=0 has been set when the standard point is in the workpiece origin.

Note 2: X axis is normally used to measure the diameter value.

4.4.12.6 Direct Input of the Tool Offset Amount

The differential value between the standard position (the tool nose of the standard tool or the tool post center) during programming and the actual used tool center is set with the following methods.

The situation of the set workpiece coordinate system



(a) Select the actual tool to machine face A in Manual mode.

(b) After pressing  button on the machine operation panel, the tool leaves face B and the spindle is stopped.

(c) Measure the distance β from the programming origin to face A.

(d) Press  for several times to switch into the interface of .

(e) Input [N] [offset number +#100] [input] to directly enter the input offset state.

Therefore, the expected page cursor of the offset number indicates the offset number, the input N is twinkling.

(f) Input [Z] + [measured value β] + [input], then, the calculated Z offset amount is input into the specified offset.

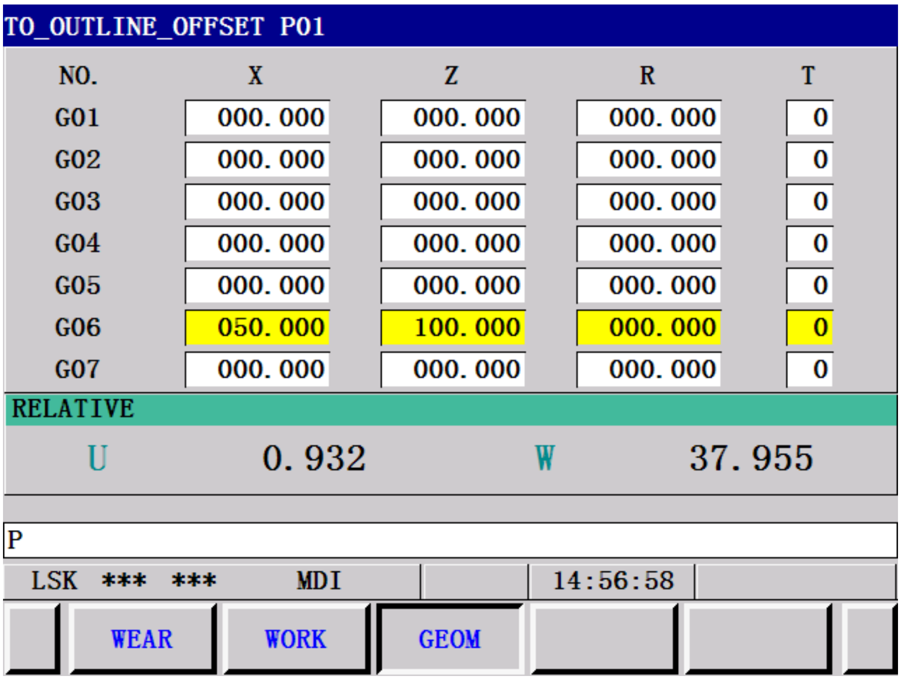
(g) Cut face B in Manual mode.

(h) After pressing  button on the machine operation panel, the tool leaves face B, and the spindle is stopped.

(i) Measure the diameter value α on face B.

(j) Input [X] [measured value α] [input], then, the calculated Z offset amount is input into the specified offset.

(k) As the above figure shows, β value is measured as 100, α value is 50, and its value is input into the 6 digits of the offset number, the method is : press OFFSET for several times, and switch into the interface GEOM, press N to input 6, Z to input 100, X to input 50, finally press INPUT, which is shown as the following figure:



Note 1: Normally, use the diameter value to measure X axis.

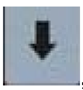
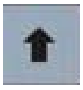
Note 2: When N or the letters on the screen lower-left corner are twinkling, the tool offset value can't be operated zero clear and incremental input. Only after inputting NO, N is cancelled or the letters on the screen lower-left corner are not twinkling, the tool offset value can be operated zero clear and incremental input with X0 or Z0.

4.4.12.7 Inputting the Offset Value of the Counter (Select Function)

During setting the offset amount, press INPUT once when only the addresses (U and W) are input without the numerical values, the value of U or W of the relative coordinate values is set.

The usage is as below:

(a) Press OFFSET for several times after switching into the interface of GEOM.

(b) Press ,  to select the interface with the offset number or input N offset number input .

(c) The standard tool moves to the standard point in Manual mode.

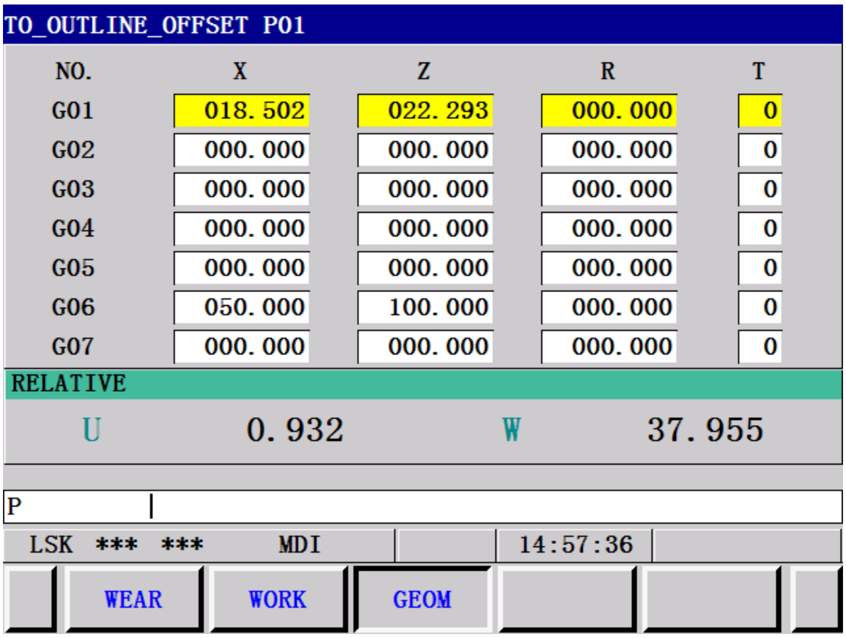
(d) Display the relative coordinate values U, W on the interface and is reset to 0. (The method is:

Press U + COR CLEAR on the relative coordinate or press SHIFT , W + COR CLEAR or press SHIFT .).

（e）Move the tool of the offset amount to the standard point. Then, the offset amount is displayed on the relative position.

（f）On the tool outline offset interface, the value of the relative position is taken as the offset amount through pressing U + input , W + input .

If it is moved to the standard point, the relative coordinate U 18.502, W 22.293 is displayed, press U to input 18.502 and press W to input 22.293, finally, press INPUT, which is shown as the following figure:



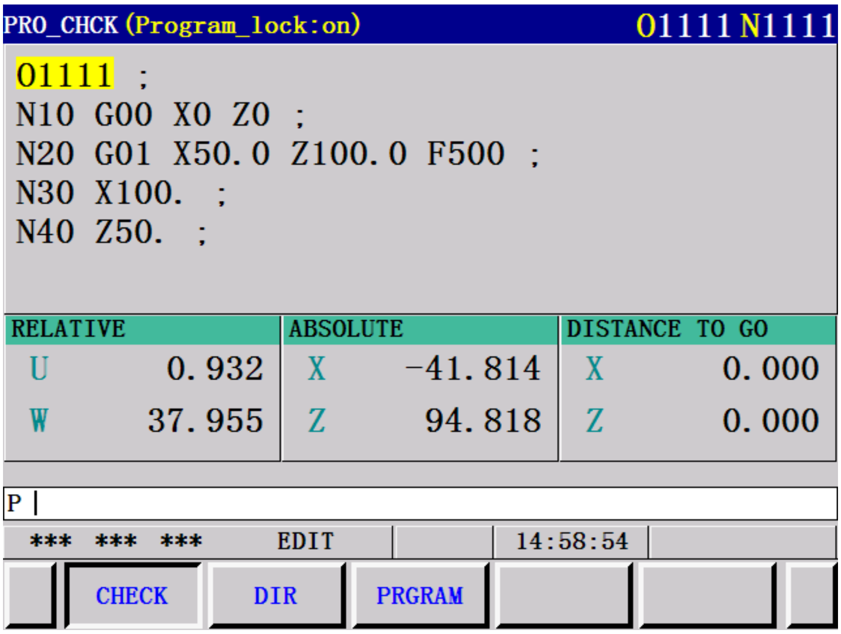
TO_OUTLINE_OFFSET P01

NO.	X	Z	R	T
G01	018.502	022.293	000.000	0
G02	000.000	000.000	000.000	0
G03	000.000	000.000	000.000	0
G04	000.000	000.000	000.000	0
G05	000.000	000.000	000.000	0
G06	050.000	100.000	000.000	0
G07	000.000	000.000	000.000	0

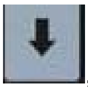
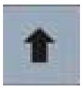
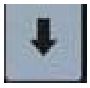
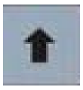
4.4.13 Display the Program

（1） Edit mode

Press PROGRAM key to enter the program display interface, there are three interfaces CHECK, DIR and PRGRAM, and they can be checked by the soft keys.



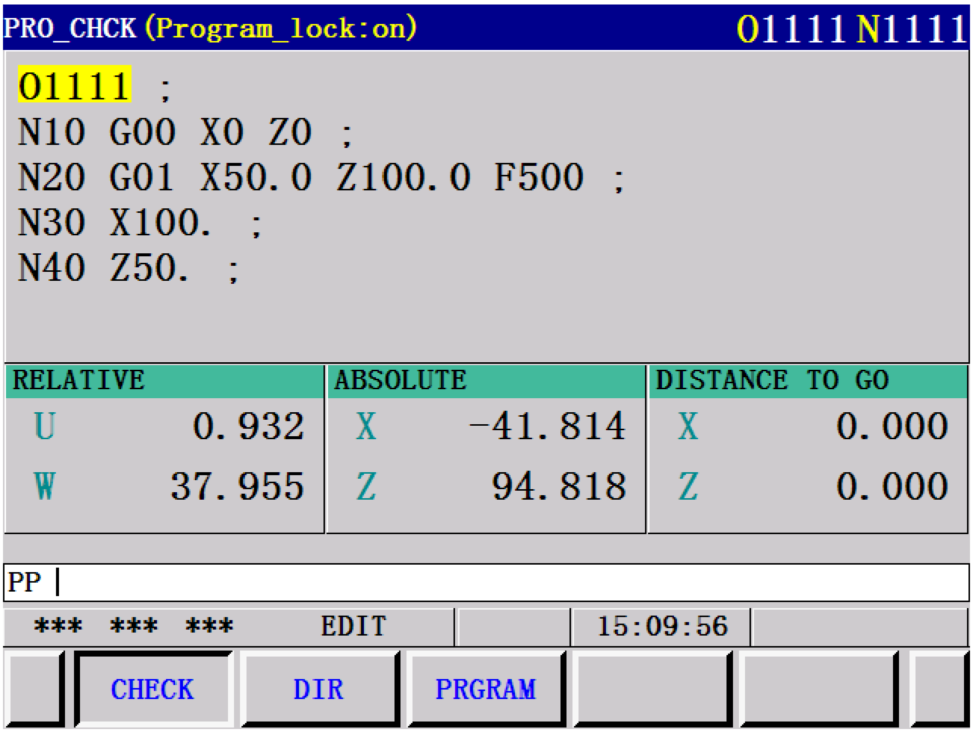
Please refer to the program search 4.4.14 for program display.

Press ,  to display the program content in sequence. Press  to display the content in the sequence；press  key to display it in the opposite sequence.

Note: **Switch into Edit mode from the other mode, and then press** **, the displayed content starts from the block being performed currently or the block already completed; if it returns to the head of the program (refer to 4.4.22.4), the program is displayed from the beginning.**

（2） Auto mode

Press  key to display the interface with the current block.

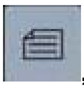
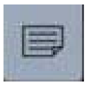
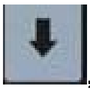
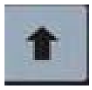


The meaning of the cursor (during automatic running):

（a） If the block is twinkling, it is going to be executed.

（b） If the block is not twinkling, it is being executed or it has already completed.

Note 1: Strictly speaking, when the buffer register is empty, if the cursor is twinkling if it is neither in Auto mode nor the feed hold, it means the next program to be executed will be read into the buffer register.

Note 2: In Edit mode, press **,**  **or** **,**  **button to move the cursor in Auto mode, the next block to be read into the buffer is the one on which the cursor is in Edit mode.**

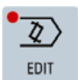
4.4.14 Searching the Program Number

When there are many programs in the memory, each program can be searched.

	O1001	O3054	O1972

⇨ Search the program number

（1） Method 1

（a）Select 

or 

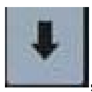
mode.

（b）Press 

key to switch into 

or 

interface.

（c）Input [O] [program number to be searched] , and press , after the searching ends, the interface with the head of the program is displayed.

（2） Method 2

（a）Select AUTO mode;

（b）Press PROGRAM to switch into PRGRAM or CHECK interface;

（c）Press O CANCEL ⬇ in order, the stored next program is displayed.

（3）Method 3

（a）Select EDIT mode;

（b）Press PROGRAM to switch into PRGRAM or CHECK interface;

（c）Press O, ⬇ in order, the stored next program is displayed.

Note 1: Return to the first program number after all the stored program numbers are displayed.
Note 2: The content of the buffer is cleared when the program number is started to search.

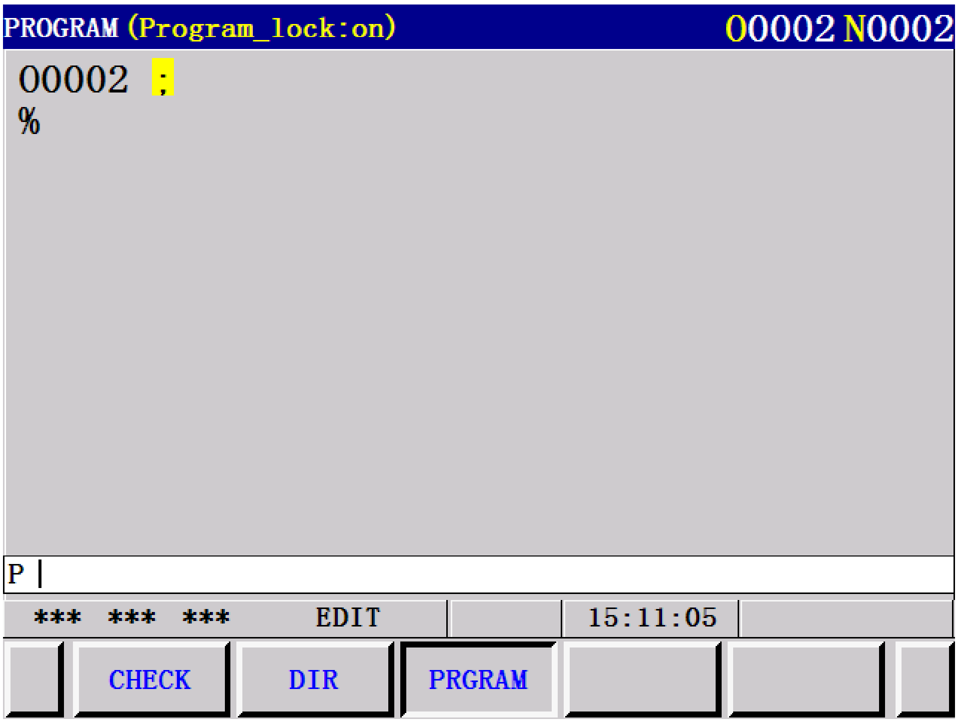
4.4.15 Inputting a Program

The machining program can be directly input from MDI keypad and stored into the memory.

（a）Switch into EDIT mode. (The program protection unlock is valid, lock is invalid).

（b）Press PROGRAM to switch into the interface of【program check】or 【program】, the current program interface is displayed.

（c）Input the program number O program number to be stored, press INSERT to change into the new interface.

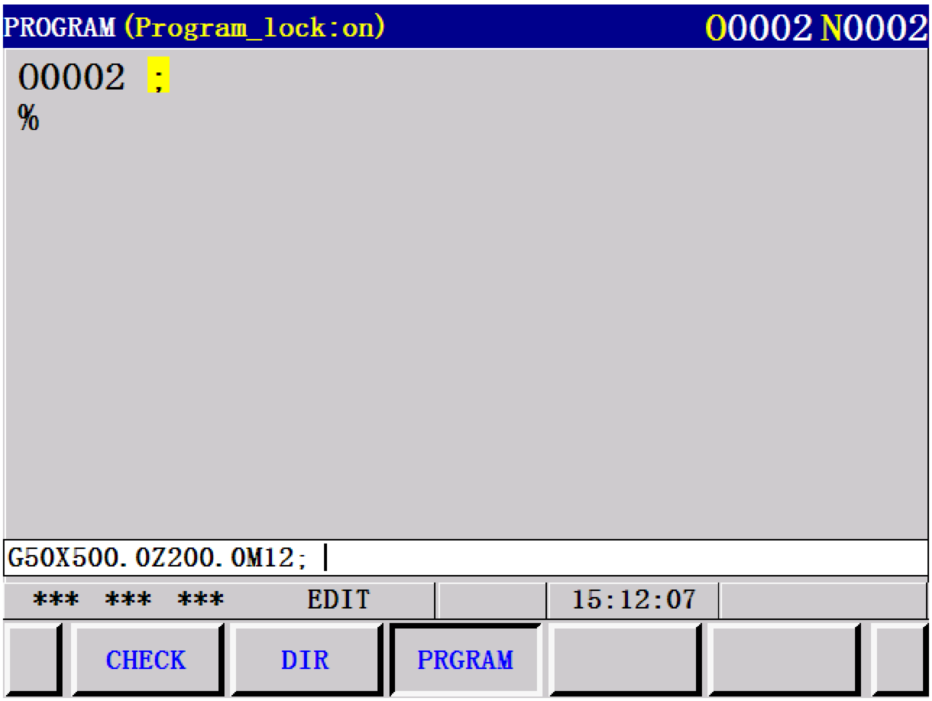


(d) Input one block based on the machining program.

Example: When G50 ×500.0 Z200.0 M12 are input

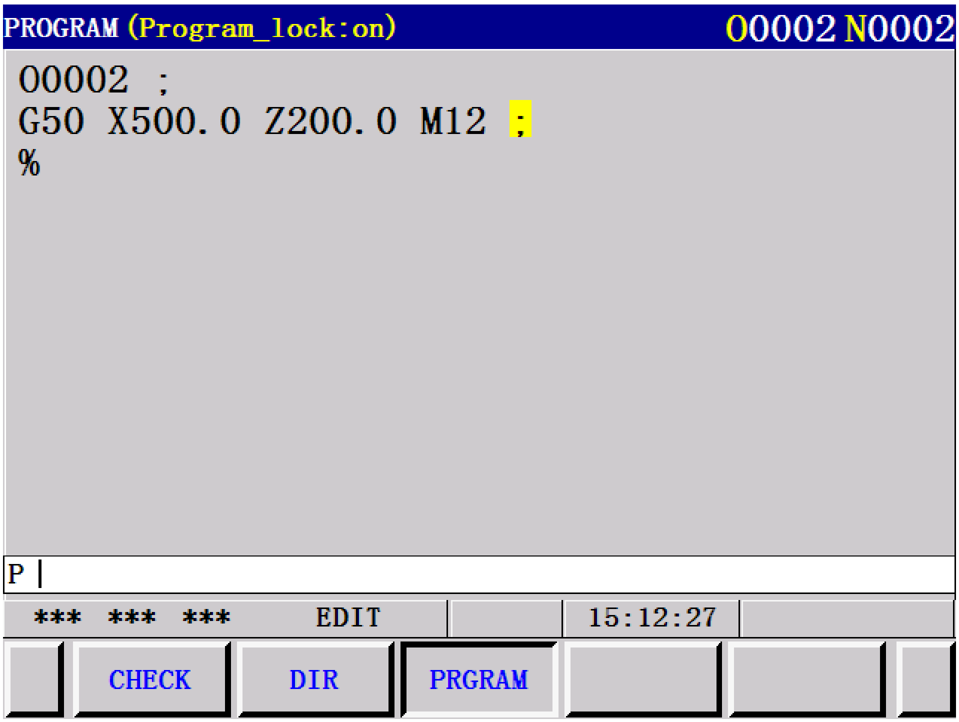
G 5 0 X 5 0 0 . 0

Z 2 0 0 . 0 M 1 2 EOB



(e) When the input is wrong, press CANCEL to clear the last input character, and continuously press CANCEL, the characters are cleared one by one from the end to the beginning. The block more than 32 characters can't be input in one time, it should be divided into several segments.

（f）After the inputting is correct, press INSERT.



（g）Input the following programs with the same method.

（h）When the input content should be rewritten, the operation is same as editing 5.30 program.

（i）At the beginning, the cursor is moved to the last input character for input continuously. The operation is exactly same as INSERT.

（j） After all input ends, the operation is completed. Press RESET button to return to the beginning.

4.4.16 Deleting a Program

The program stored in the memory should be deleted (When the program protection unlock is valid,lock is invalid);

（a）Select EDIT mode.

（b）Press PROGRAM, it will be switched into any interface of 【program check】or 【program content】or【program】

（c）After inputting O program number , press DELETE, the program of the input program number will be deleted.

4.4.17 Deleting All Programs

When all programs stored in the memory are deleted (the program protection unlock is valid, lock is invalid):

(a) Select Edit mode.

(b) Press 

to enter one of the interfaces 【program check】or【program content】or【program】.

(c) Input O - 9 9 9 9, and then press 

.

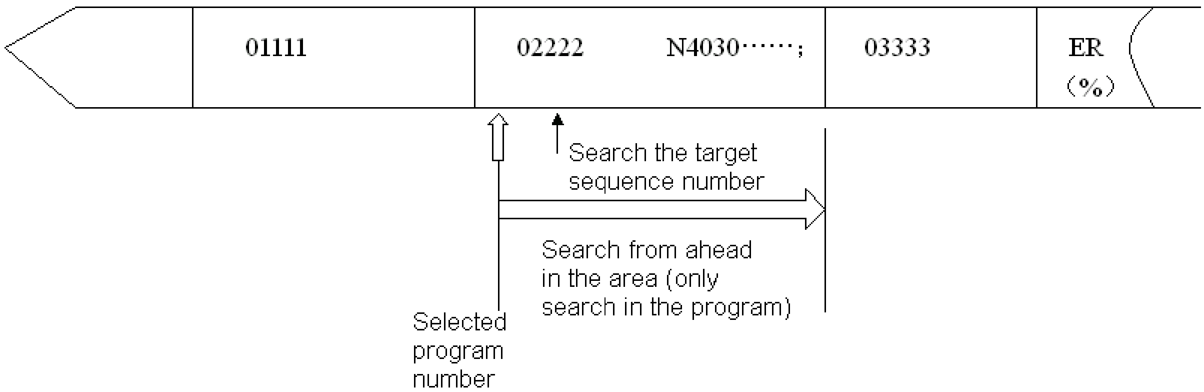
4.4.18 Searching the Sequence Number

Searching the sequence number is to search one sequence number in the program, and its purpose is to start from the block of the sequence number or execute it, again.

During searching, the skipped block doesn't affect NC, that is to say, the coordinate value of the skipped block and M, S ,T and G codes won't change NC coordinate value and the modal value. When the user macro is selected, the sequence number N during research isn't displayed.

Therefore, at the beginning of searching the sequence number or at the first block to be executed again, specify M, S,T and G codes to set the coordinate system. The block searched by the sequence number is always one break point in one process.

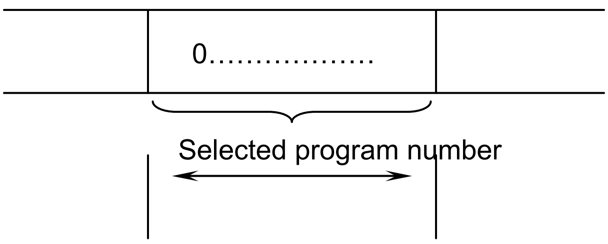
To start the block searched in the program , it's necessary to check the machine and NC states, specify M, S,T and G codes to set the coordinate system.



(a) Select 

mode.

(b) Select the program number of the sequence one to be searched.



Only search the range

When the program includes the sequence number to be searched, operation（c）is executed. However, if the sequence number to be searched is not in the program, the program number of the sequence one to be searched is selected with the program number search.

（c）Press PROGRAM to switch into the interface of PRGRAM or CHECK .

（d）Input N sequence number to be searched , press ⬇ to search the sequence number.

Note 1: During the indexing, the coordinate value and the modal data won't be changed. After searching ends, command MDI data if it's required.

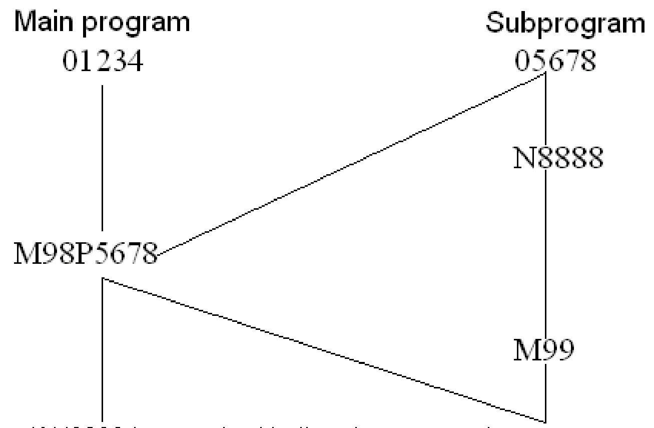
Note 2: Check during searching:

Select the block to be skipped

Alarm detection（03，04，05，10）

Note 3: During the program searching, only when the program with number "N××××", the searching can be operated; otherwise, the alarm is issued.

Note 4: M98P×××× (calling the subprogram) is not executed during searching the sequence number; and in Auto mode, the alarm（NO.060）occurs if the sequence number is searched in the subprogram called by the currently selected program.



The alarm occurs if N8888 is searched in the above example.

4.4.19　Restarting a Program

When the tool gets damaged or the machine restarts after stopping machining, the block is restarted from the specified sequence number with the function.

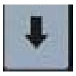
（1）The tool gets damaged (type P).

（a）Press FEED HOLD button, tool retraction, change into a new tool , the offset amount is rewritten when it's required.

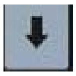
（b）Set the machine operation panel PROG. RESTART as ON.

（c）Press PROGRAM button to display the current program.

（d）Return to the start position of the program (In automatic operation, press ⬆ button).

(e) Input [P] [sequence number], press  and the following figure occurs.

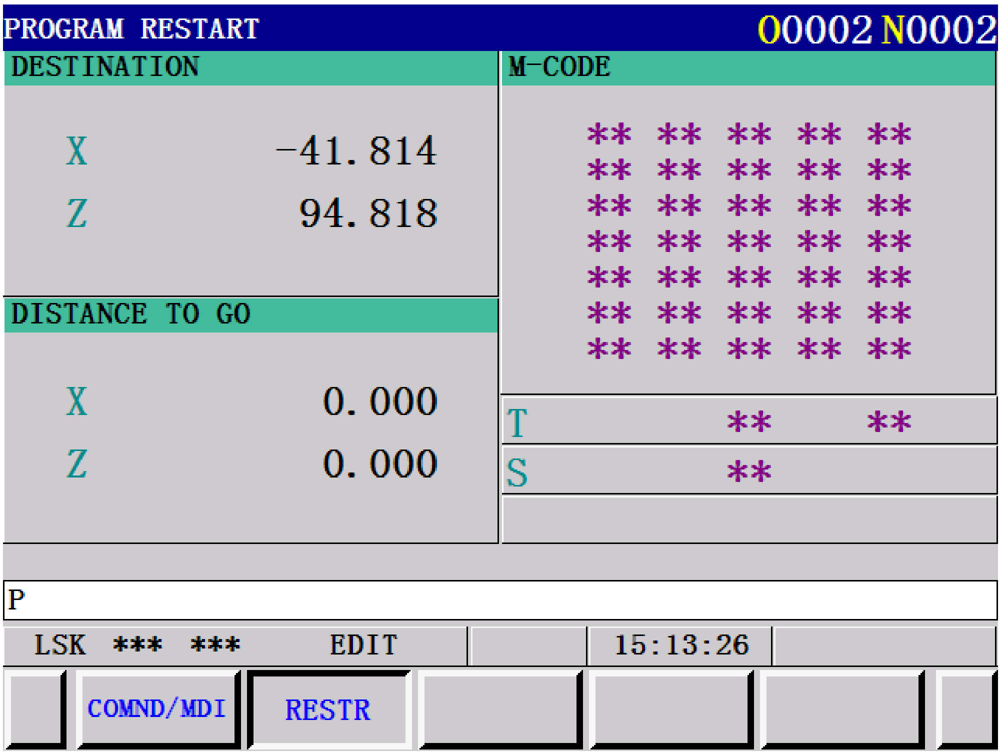
Search the block to be restarted, when the same sequence number occurs for many times, (for example, searching the sequence number in the subprogram called for many times), and it's stipulated that the ahead four digits are the time which the searched sequence number occurs in the block, and the last four digits are the sequence number.

P [1 2 3 4] [0 1 2 3] Press 

Times sequence number

When the time is 1, the ahead four digits are omitted, and the leading zero of the other sequence numbers can be omitted except for the specified times.

(f) After searching ends, LCD screen displays the program restarting interface.



DESTINATION in the figure displays the position of restarting machining; DISTANCE TO GO displays the distance between the current tool position and the position of restarting machining.

M is represented as 35 commanded M codes.

T is represented as 2 commanded T codes.

S is represented as the last commanded S codes.

At the starting position of M or T codes, it displays the first commanded code.

(g) Press  button, it is OFF.

(h) Observe the figure, output M, S and T codes with MDI output in MDI mode. In such situation, M, S and T codes to be output aren't displayed on the program restarting interface.

(I) For the automatic return in Auto mode, check whether the distance displayed on the remaining distance is correct or not, and whether the tool hits the workpiece when the tool is

moved to the machining restarting position. In Jog mode, the tool is moved to one position on which the following movement doesn’t hit the workpiece, and then press the cycle start button. Then, the tool moves to the position of restarting machining in manual feedrate based on the sequence of Z and X axes, and “RESTR” disappear; and the machining starts from the block of the program restarting.

（2）The machining (type Q) is restarted after the following situation occurs

（a）Cut off the power supply

（b）Press the emergency stop button.

（c）The machine instantly stops due to the stored stroke limit alarm.

（d）The coordinate system is changed after the previous automatic operation.

Example:

（Ⅰ）G50 command is given by MDI.

（Ⅱ）The coordinate system moves.

（Ⅲ）The automatic coordinate system is set after the reference position return.

（Ⅳ）The coordinate system zero clear is operated.

（a）After power on or the emergency stop is released, or the stroke limit alarm is released, the reference position return is operated before the machine is restarted.

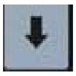
（b）The tool is moved to the program machining start position in Jog mode, and set the modal data and the coordinate system same with those of states of the machine restarting.

（c）Set or change the offset amount if it’s required.

（d）Set the machine operation panel  as ON.

（e）Press  to display the program. Search the program if it’s not the required one.

（f）The program is returned to the start position (in Auto mode, press ).

（g）Q sequence number, press , and the interface is switched into the program restarting one.

Search the restarting block of the sequence number.

During searching, when the same sequence number occurs for many times, it’s stipulated that the ahead four digits are the times which the searched sequence number occurs in the block, and the last four digits are the sequence number.

Q 1 2 3 4 0 1 2 3 Press button 

times sequence number

When the time is 1, the ahead four digits are omitted, and the leading zero of the other sequence numbers can be omitted except for the specified times.

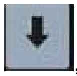
（h）After searching ends, LCD screen displays the program restarting interface.

（i）Check whether the tool hits the workpiece when the tool is moved to the machining restarting position. If it hits, move the tool into the position to avoid hitting.

（j）Check whether the distance is suitable in the remaining distance.

（k）Return to the Auto mode. press the cycle start button. Then, the tool moves to the position of

restarting machining in manual feedrate based on the sequence of Z and X axes.

Note 1: In the following conditions P sequence number , the program restarting is not executed.

（Ⅰ）The automatic operation is not executed after power on.

（Ⅱ）The automatic operation is not executed after the emergency stop is released or the stroke limit alarm in the stored type is not performed.

（Ⅲ）The automatic operation is not executed after setting, changing or moving the coordinate system.

P/S 97 alarm occurs caused by the above conditions （Ⅰ），（Ⅱ） or resetting after 94，96 or 97 alarm.

Setting the coordinate system results in P/S 94 alarm.

Moving the coordinate system results in P/S 96 alarm.

The block of restarting machining is one of many blocks, it is after the block of which the last set or changed coordinate system before the machining interruption.

Note 2: In P and Q modes, when the tools are respectively moved to the machining restart position in single axis each time. When the 1st axis movement is completed, the single block stop is possible, while MDI operation can't be inserted, but manual operation can be inserted; the returned axes can't be moved.

Note 3: During searching, when the conditions of input signal and the offset amount, etc are different with the previous ones, the tool can't return to the correct machining start position. Even the single block switch is ON or switched, the searching continues in MEMCRY/TAPE mode.

Note 4: During searching, when the feed hold is valid, the resetting is operated during or after searching, the program is restarted from the beginning. While the searching ends, the resetting state is set by parameter 006，CLER in MDI mode.

Note 5: When the program restarting switch is ON, the cycle start is ignored.

Note 6: No matter before or after machining, the manual absolute switch is always ON during the manual operation.

After the manual operation is executed, one program is restarted rather than reset; or the manual operation is executed along the axis which hasn't been returned to the machining restart position, it's assumed that the manual absolute switch is ON no matter whether the manual absolute switch is ON or OFF.

Note 7: In the following situations, the tool can't be returned to the correct position in principle.

（a）The manual absolute switch is OFF during the manual operation.

（b）The tool is moved when the machine is locked.

（c）The external mirror image function is used.

（d）The coordinate system isn't set at the start position of the incremental programming.

（e）The program mirror image function is used.

（f）The manual operation is inserted during the axis returning.

（g）In the machine lock state, the machine is unlocked after the program is restarted.

（h）The program restarting command is sent when the block of the skip cutting or the block before the one of the absolute command is executed.

（i）The restarting program is commanded when the combined block of the canned cycle is executed.

（j）After searching ends, the coordinate system is set or offseted. But in case of (c), the tool return can be operated in P mode in the block after the one of switching ON/OFF. Then, the same state is hold in the mirror image interruption. Moreover, in case of (f), the tool return can also be operated in P mode

Caution, no alarm occurs in the above situations!

Note 8: No.60 alarm occurs when the specified program only includes M98,M99, the macro program calling commands （G65，G66，G77）or the macro program sentences, or the block not stipulated is searched.

Note 9: P/S alarm（98）is issued after the power is on or after the emergency stop is released or the stroke limit alarm (stop immediately) is released and the program restart is commanded and G28 is detected while the reference position return is not executed.

Note 10: P/S alarm（99）after the program restart searching ends, the axis movement is executed in MDI mode.

Note 11: "RSTR" is twinkling at the bottom of LCD screen after the program restarting is commanded and before the last axis (Z) is returned.

4.4.20 Sequence Number Comparison and Stop Function

The function is to stop machining after the commands are executed into the preset sequence number.

（a） Select 

mode.

Press setting button to switch into the setting interface (press ↓ to move the cursor to the setting number 180, and the address N can't move the cursor.).

（b） Press P sequence number to stop and INPUT to input commands in order.

（c） Select AUTO mode to make the machine ready for automatic operation.

（d） Press CYCLE START button. The machine stops after executing the data in the block with the preset sequence number based on step (c).

（e） The preset sequence number is cleared during executing the program comparison and stop meanwhile.

（f） If it requires to execute the comparison and stop once more, repeat the steps from (a).

Note 1: The sequence number 0 can't be taken as the sequence number comparison and stop.

Note 2: The preset sequence number can be cleared by resetting. Therefore, automatic running is performed after setting without resetting.

4.4.21 Display Parameters

Press PARAMETER, the parameters can be displayed. Because there are many interfaces to display parameters, press  or  to display the parameters of each interface. About the meaning of parameters, refer to Appendix 5.

4.4.22 Editing a Program

The program content stored in the memory can be rewritten, and the program word is rewritten based on the unit.

（1）Select EDIT mode.

（2）Press PROGRAM.

（3）Select the program number. If the program has been already chosen, direct operation from the step (4); otherwise, search the program number.

（4）Search the word to be rewritten.

（a）Scanning.

（b）Searching with word.

（5）Rewrite, insert or clear the word.

Note 1: Concept of the word and edit unit

The word is composed by the address and the following digits. Because the concept of the word is not clear regarding to the user program, the concept of "editing unit" is used. Editing unit is the object to rewrite or clear. During scanning for one time, the cursor is moved to the head of the editing unit, and the word can be inserted behind the editing unit.

The definition of the editing unit:

（1）Before one address switching into the other one.

（2）Address: character, IF，WHILE，GOTO，END，DO，=，;（EOB）.

Based on the definition, one word is also an editing unit.

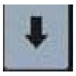
About the explanation of editing, strictly speaking, the word should be called as the editing unit.

Note 2: During the program editing, the machining pauses because the single block stop or feed hold, the program is not allowed to execute after the program rewriting, inserting and clearing; otherwise, the program can't be performed correctly. The program content is displayed on LCD after it is continued executing. Therefore,

resetting should be operated before rewriting the content in the memory with the edit function; or resetting is executed after editing, and then the program is executed.

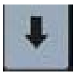
4.4.22.1 Scanning

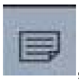
Scan one word each time.

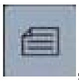
（1） Press 

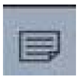
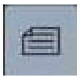
Then, the cursor is moved ahead along the characters on the screen. The cursor points below the address character of the word.

（2） Press , then, the cursor is moved based on each character in the opposite direction on the screen. The cursor points below the address character of the word.

（3） Press  or  all the time, the searching can be operated continuously.

（4） Press , the next page is displayed. The searching is started from the head of this page.

（5） Press , the previous page is displayed. The searching is started from the head of this page.

（6） Press  or  all the time, display page by page.

4.4.22.2 Method of Searching a Word

The method is used during searching the word.

N1234×100.0 Z125.0； S1234； N5678 M03

Searching/scanning word
(displayed word)

To search S1234

Searching direction →

（1）Input S 1 2 3 4

Note 1: Just input S123 from the keypad, S1234 can't be found.
Note 2: When S009 is searched, just inputting S9 can't be found, so S009 must be input.

（2）Press  to start searching.

After searching ends, the cursor points below S of S1234.

4.4.22.3 Method of only Searching an Address

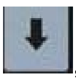
From the present position, the specified address is searched in sequence.

N1234 X100.0 Z125.0； S1234； N5678 M03

Searching/scanning word
(displayed word)

To search M03

（1）Input M

（2）Press , searching is started, and the cursor points below M letter.

Note: ↑ can't be used for searching word and address.

4.4.22.4 Method of Returning to the Program Head

01100	N0001×12.34；	Z15.67；	G01X12.5；	M04

↑ Beginning

↑ Searching /scanned word (displayed word)

（1） Method 1: In EDIT mode, press RESET, the program is displayed from the beginning.

（2） Method 2: Search the program number.

（3） Method 3:

（a）Set in AUTO mode.

（b）Press the function button program to switch into the interface of PRGRAM.

（c）Press ↑, the program is displayed from the beginning.

4.4.22.5 Inserting a Word

To insert T105 ↓

N1234×100.0 Z125.0； S1234； N5678 M30；

↑ To search

（1）Search or scan one word before the word to be inserted.

（a）Refer to 4.4.22.1 during scanning.

（b）Refer to 4.4.22.2 during searching the word

The cursor is on the place of Z1250.

（2）Input the word to be inserted.

T 1 0 5 press INSERT.

Note 1: When the data are inserted without the address, insert the data into the present word (editing unit); on the above example, search Z1250, the cursor is below Z; once 2.5 is inserted, the word becomes Z12502.5.

Note 2: Same, after the numerical value is added to all addresses of EOB，LF, insert 23 below "；" on which the cursor points, and the word becomes 23, but it isn't significant in the program.

4.4.22.6 Rewriting a Word

N1234×100.0 Z125.0 T105; S1234

To change into M15

(1) Search or scan the word to be rewritten.

(2) Input the word to be rewritten.

M 1 5 ALTER.

N1234×100.0 Z125.0 M15; S1234

The changed content

4.4.22.7 Inserting or Rewriting Many Words, Blocks or Character String

Many words, blocks or character string can be inserted, but maximum 32 characters can be inserted one time.

Insert T105 M20 in the previous example, input:

T 1 0 5 M 2 0 INSERT

Same, the word pointed by the cursor can be rewritten to many words, blocks or character string.

Note 1: When the cursor points below Z of Z125.0, the inserted 2.5 M20 becomes Z125.02.5 M20.
Note 2: When the cursor points below T of Z125.0 T105, the inserted 2.5 M20 becomes Z125.02.5 M20.

4.4.22.8 Clearing a Word

N1234×100.0 Z125.0 T105; S1234;

To clear Z125.0

(1) Search or scan the word to be cleared;

(2) Press DELETE;

N1234×100.0 T105; S1234;

Content after clearing

4.4.22.9 Stop Clearing before EOB

Searching /scanning word (displayed word)

N1234×100.0 Z125.0 T105 M13; S1234;

Clear the area

Press EOB, DELETE in sequence, and delete the content before "；", and the cursor is moved to the first word of the address in the block to be cleared.

S1234

4.4.22.10 Clearing Many Blocks

Clear the content from the present displayed words to the block of the specified sequence number.

N1234 M10； M15 ×10.0；…… T0122； N2233 S1200； N3344 Z10.0；

Clear the content in the range

（1）From the cursor starting position, input the sequence number of the last block which is one of many blocks to be cleared.

N 2 2 3 3

（2）Press DELETE.

N3344 Z10.0；

4.4.22.11 Arranging a Memory

Because the part program should be often edited, it causes the unreasonable usage for the memory and the data with the specified length can't be stored. Therefore, the memory should be arranged.

（1）In EDIT mode.

（2）Press PROGRAM to switch into the program or the program check interface.

（3）Press CANCEL and SHIFT in order.

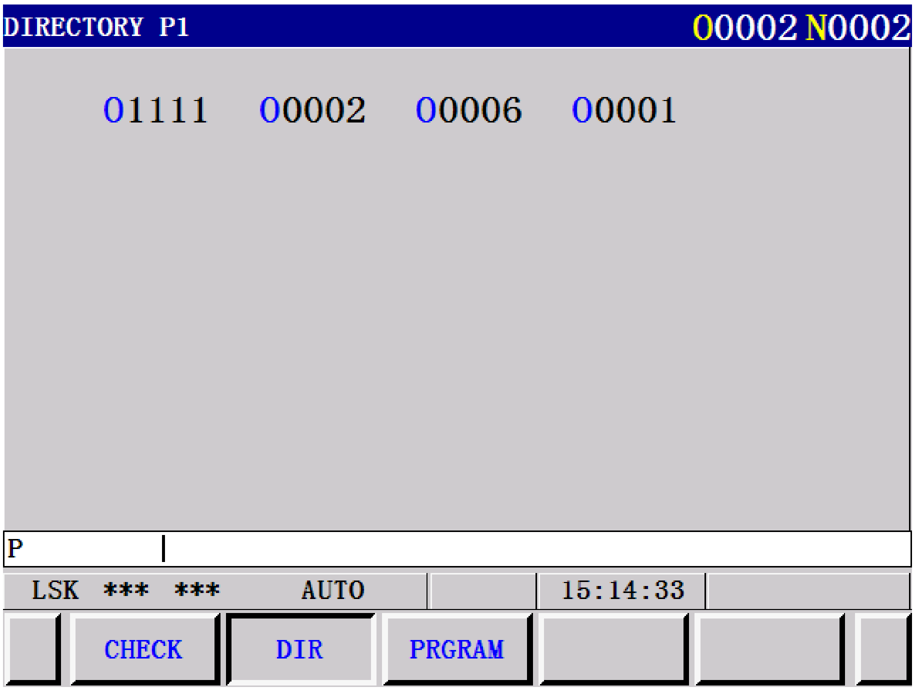
After arranging, the number of storable characters is displayed at the left bottom on the program content screen.

Note 1: When there is only one program in the memory, it is stored on the standard length; if many programs, recognizing these program will occupy some storage space.

Note 2: To accelerate the program editing speed, during rewriting and inserting, the remaining memory space left by the actual rewritten or inserted characters may be wasted, while arranging the memory can eliminate the wasting.

4.4.22.12 Display All Program Numbers of All Programs in the Memory

As introduced in 5.22.11, once the content of the memory is cleared, all the program numbers of the stored programs are displayed.



4.4.22.13 Editing a User Macro

The editing of the user macro is operated same as editing a program in Edit mode which has already released the program protection state, but they also have the different points.

（a）Press 

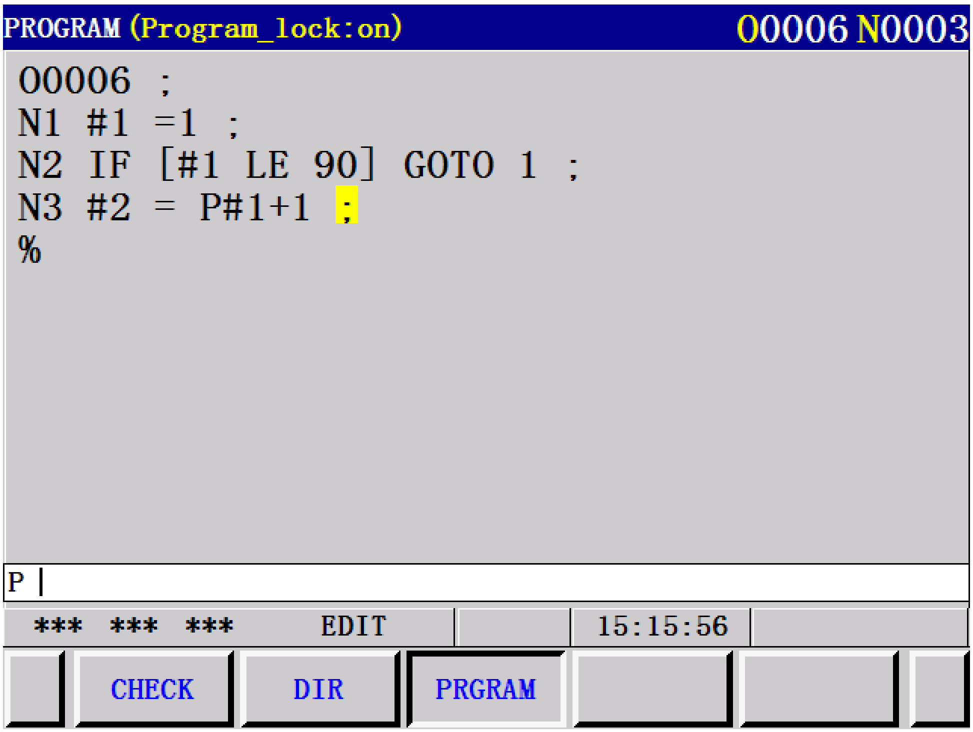
.

Press 

once, the cursor is changed from "—" used commonly into " ^ ". In such state, press the buttons of digits or letters from MDI keypad, then, the corresponding digits or letters are input.

After one character is input, the cursor is restored to "—"; if the displacement key is pressed twice, the cursor also returns to "—".

Example:



(b) Deleting, rewriting and inserting a program

When the input user macro is required to be edit, the cursor can only move into the following characters or the marks.

(Ⅰ) Address

(Ⅱ) Select block skip /

(Ⅲ) Replace # at the beginning in the left of the sentence

(Ⅳ) (, =,;

(Ⅴ) The initial character of IF, WHILE, GOTO, END, DO, SETVN.

On LCD screen, the space of one character is left before the above characters and marks. Deleting, rewriting and inserting the program is operated based on the unit which is the characters between two cursor positions.

(Example) The position of the cursor

N001 X-#100 ;

#1=123 ;

N002 / 2 X [12/#3] ;

N003 X-SORT [#3/3*#4+1]] ;

N004 X-#2 Z#1 ;

N005 #5 = 1+2-#10 ;

IF [#1 NE 0] GOTO 10 ;

WHILE [#2 LE 5] DO 1 ;

[2000+#2] = #2*10 ;

#2 = #2+1 ;

END 1 ;

Note 1: The cursor can't stop in the bracket ( ).

Control pause/restore

(Example) (#1 = 100) ;

No stopping in the position

Note 2: The position of the cursor varies as the program is rewritten.

(Example) Z200 of X 100 Z200 ; is rewritten into 100 with 

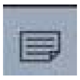
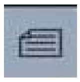
, it's changed into X 100100;

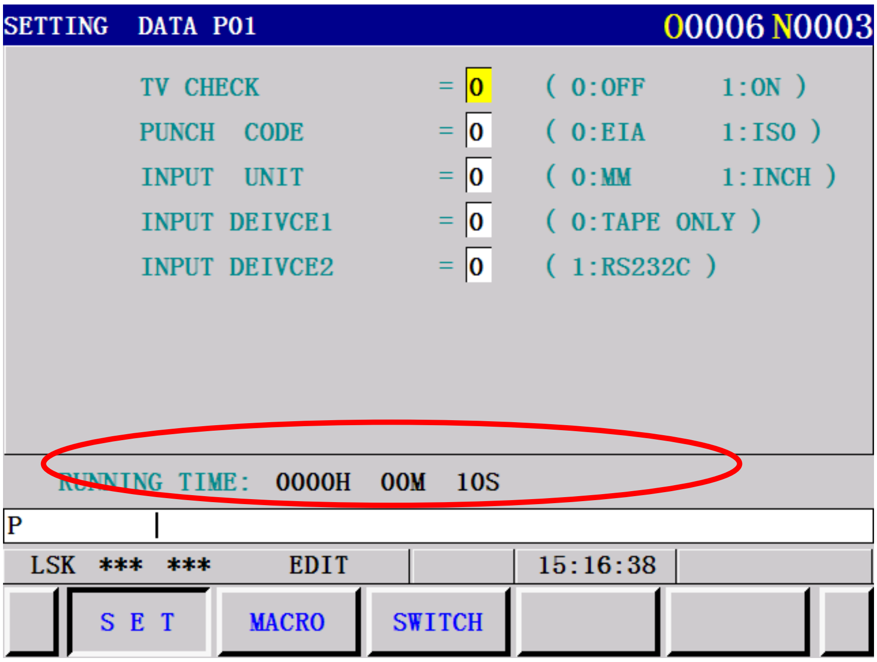
(c) Abbreviation of the custom macro

When the custom macro is rewritten or inserted, it can be replaced by the initial two letters. That is to say: WHILE, GOTO, END, SOR, AND, SIN, COS, TAN, ATAN, SORT, ABS, BCD, BIN, FIX, FUP, ROUND, SETVN can be replaced by the marked part.

(Example) The input data WH [TA [#1*AB [*2]] LERO [#3]] are same with WHILE [TAN [#1*ABS [#2]] LE ROUND [#3]].

4.4.23 Display Running Time

After the automatic operation time is accumulated, it is displayed on LCD screen on unit of hour, minute and second (the unit two seconds.) Based on setting , the time shown on the following figure is displayed. When the system is on the other interface, press  or  to switch.



Note 1: The accumulative time is the running one excluding the single block stop and feed hold time.

Note 2: After the automatic operation stops, the power supply is cut off, and the maximum error is 6 minutes after power on, again.

Note 3: The time can be preset through setting operation and the data numbers are 57, 58, 59, please refer to 4.4.8.

4.4.24 Menu Switch Function

Use LCD to replace the switch on the machine operation panel to power on/off the CNC memory, and the switch quantity on the machine operation panel can be reduced through the function.

The signals of LCD switch are as below:

（1） SINGLE BLOCK
（2） MACHINE LOCK
（3） DRY RUN
（4） BLOCK SKIP1～9
（5） X MIRROR IMAGE，Z MIRROR IMAGE
（6） DISPLAY LOCK
（7） Auxiliary function lock（AUX FUNC LOCK）
（8） Manual absolute value（MANUAL ABSOLUTE）

Once the switch signal is stored in the memory on LCD, it doesn't change even at power off.

These signals are not only set by LCD, but also by the corresponding buttons on the operation panel.

The signals are taken as connected no matter the signals from the machine operation panel or LCD.

Therefore, any signals are cancelled or cut off from the machine operation panel, the signals can still be switched with LCD.

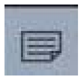
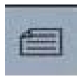
Refer to the machine operation panel for MDI/LCD switch interchange.

（1） Setting and display

The states of above signals on LCD can be displayed with the following operations.

（a） Display

（Ⅰ）Select the 3rd interface for setting, firstly press SETTING for setting, and then press SETTING twice.

（Ⅱ）Because there are two interfaces, select the required interface with  or  buttons.

The 1st page: Display the selected switches except for the block skip ones.

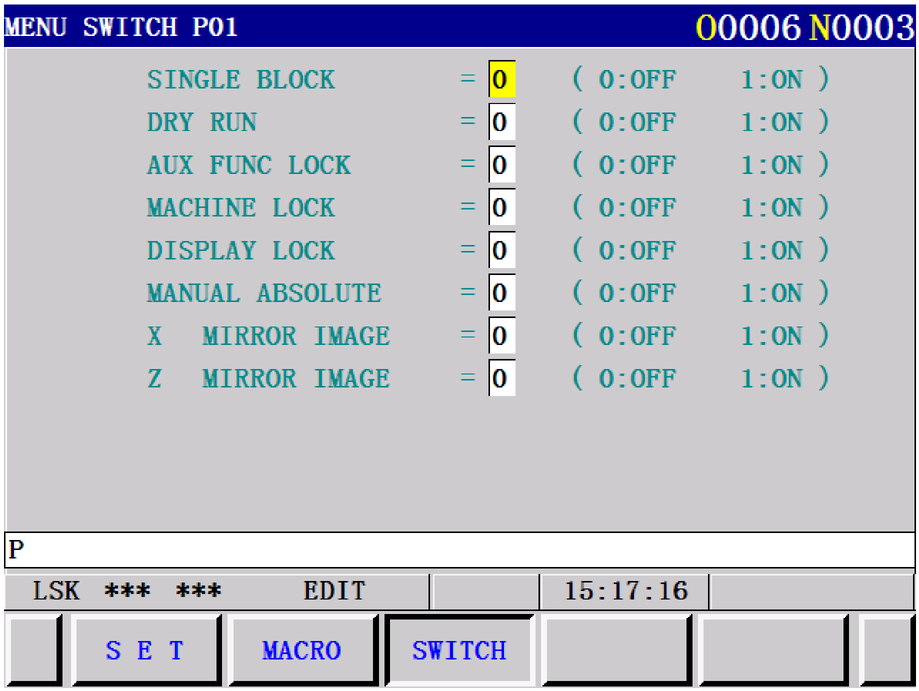
The 2nd page: Display the selected switches except for the block skip 1~9 ones.

（b） Setting

The following operation is performed after display

（Ⅰ）Press  or  to move the cursor to the item to be changed.

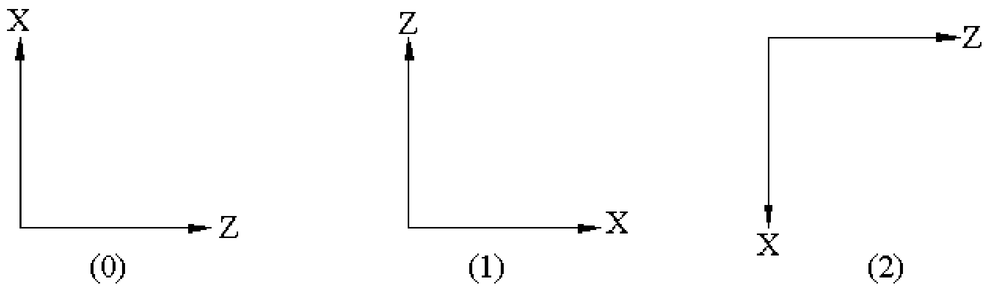
（Ⅱ）After pressing the address P , insert 1 during connecting, insert 0 during disconnecting. Input P , 0 or 1 , input.



4.4.25 Drawing Function

On LCD screen, the tool path of the program during machining can be drawn, so the process of actual machining can be checked. Moreover, the figure can be scaled up or down.

The following two drawing coordinate systems can be set by parameters.

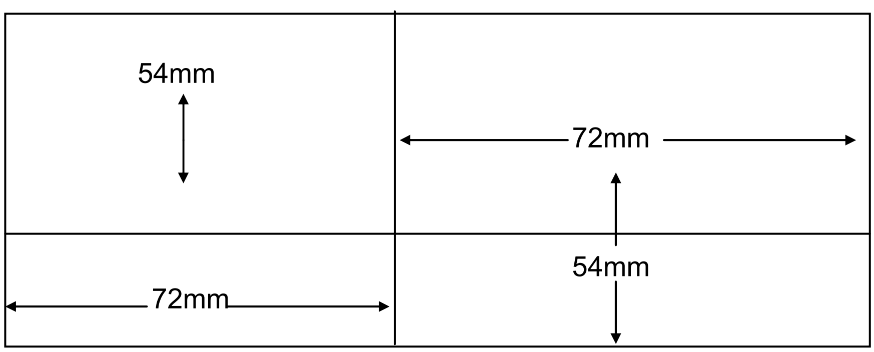


AXIS PLAN SELECT=0 select（1）

AXIS PLAN SELECT =1 select（2）

AXIS PLAN SELECT=2 select（3）

The dimension of the maximum drawing (on LCD) is as below:



On the above figure, the maximum horizontal range and the vertical one are respectively 144mm and 108mm. If the programmed figure range is more than the specified one, the drawing scaling is applied. The override range is 0.01～100.00 times, usually, the override is set by the following factors.

Drawing override= the smaller value of H or V in the drawing override.

$$\text{Drawing override H}=\frac{\alpha}{\text{Programmed path length in horizontal direction}}$$

$$\text{Drawing override V}=\frac{\beta}{\text{Programmed path length in vertical direction}}$$

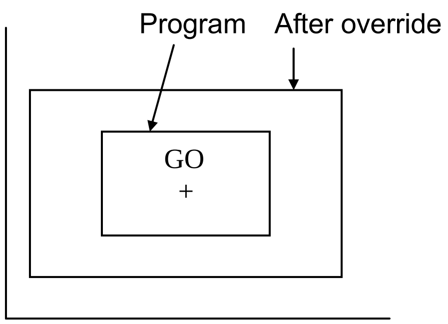
α ：144mm（specified by radius）

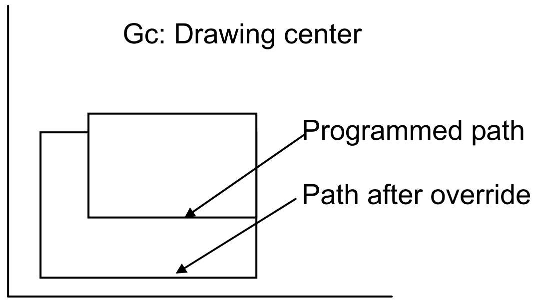
288mm（specified by diameter）

β：108mm（specified by radius）

216mm（specified by diameter）

Note 1: Z axis is normally specified by the radius and X axis is specified the radius/diameter. The scaling is generally relative to the drawing center.





When the tool path is not closed to the drawing center, the tool path will exceed the drawing range after scaling the figure. Then, the following four parameters can be used:

RANGE X （MAX）: Range X（Maximum）

RANGE Z （MIN）: Range X（Minimum）

RANGE Z （MAX）: Range Z（Maximum）

RANGE Z （MIN）: Range Z（Minimum）

Then, the drawing center（GCX，GCZ）can be calculated by the following formula:

GCX=[X（MAX）+X（MIN）]/2

GCZ=[Z（MAX）+Z（MIN）]/2

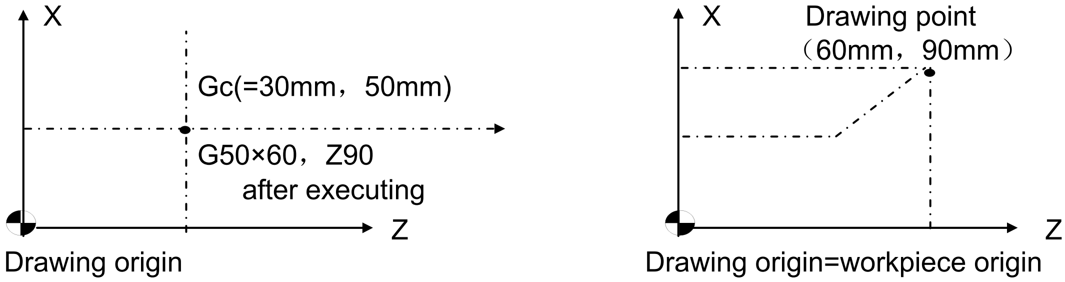
Note 2: Even X axis is specified by the diameter during programming, X（MAX），X（MIN）is also specified by the radius.

The values are set based on the input units 0.001mm or 0.0001mch. Moreover, if “the inserted unit 1/10” is selected, it is changed into 0.0001mm or 0.00001inch.

As long as these parameters are set, they remain valid during selecting the drawing before pressing T key.

Even the workpiece coordinate origin is changed, the drawing origin and center doesn’t change on the figure. In other words, the workpiece coordinate origin is always complied with the drawing one.

（Example）



After executing G50 command, the drawing point is moved as the dot and dash line（- - - - - ·）.

For the rapid traverse, the tool path is indicated as the dot line (··················), for the cutting feed movement, the tool path is indicated as the real line (——— ）

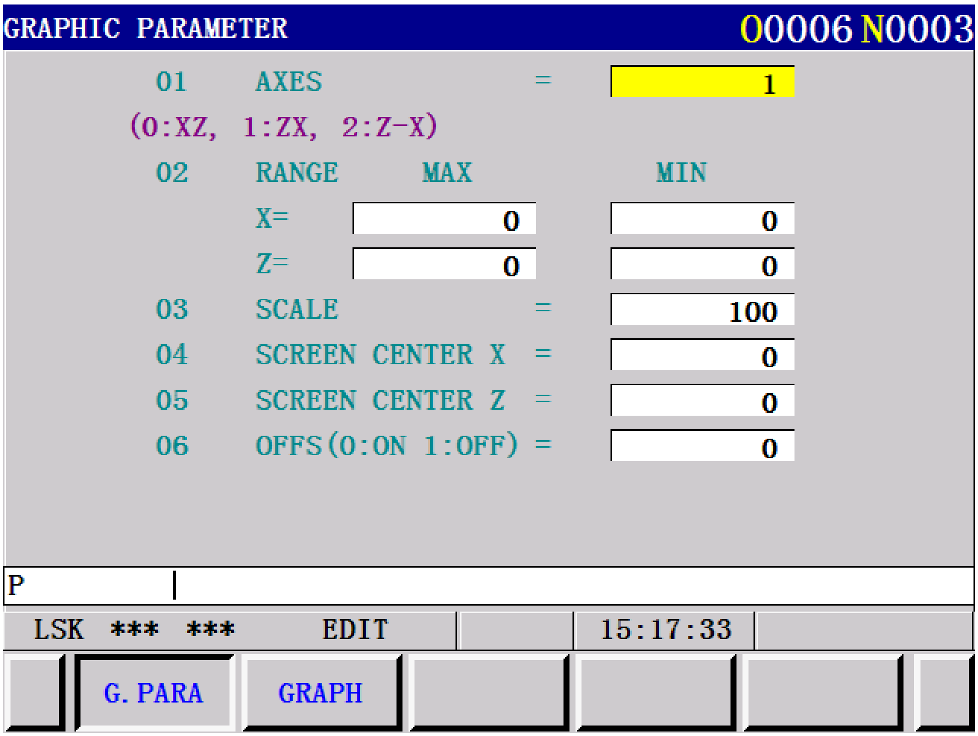
4.4.25.1 Operation

（1）Setting the number of drawing

For drawing, various parameters must be preset, and they can be set no matter in any mode.

a) Press 

, the following drawing parameters are displayed on LCD.



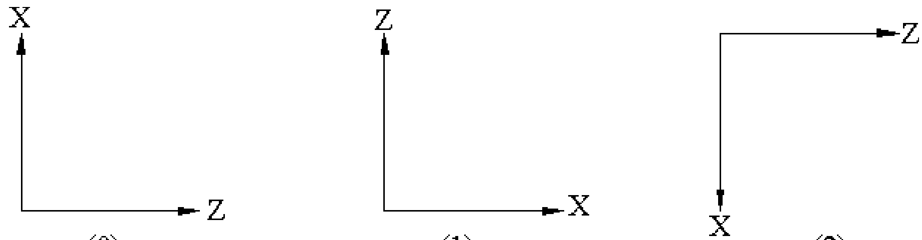
② Press  or  key, the cursor is moved to the target position.

③ Operate with P numerical value and input, the drawing parameters are set.

④The steps ②③ are repeated to set the required numerical values.

（2）Note of the drawing parameters.

AXIS PLAN SELECT: Set the drawing plane （XZ=0, left ZX=1and right ZX=2）



(0) (1) (2)

RANGE（MAX），（MIN）: In the range, set the maximum and the minimum values of the tool path in the figure.

X=0，0

Z=0，0

Setting range:

0～±99999999

1=0.001mm or 0.0001inch（normally）

1=0.0001mm or 0.00001inch（The input unit/10）

Setting range:

1～100000

1=0.01 time（normally）

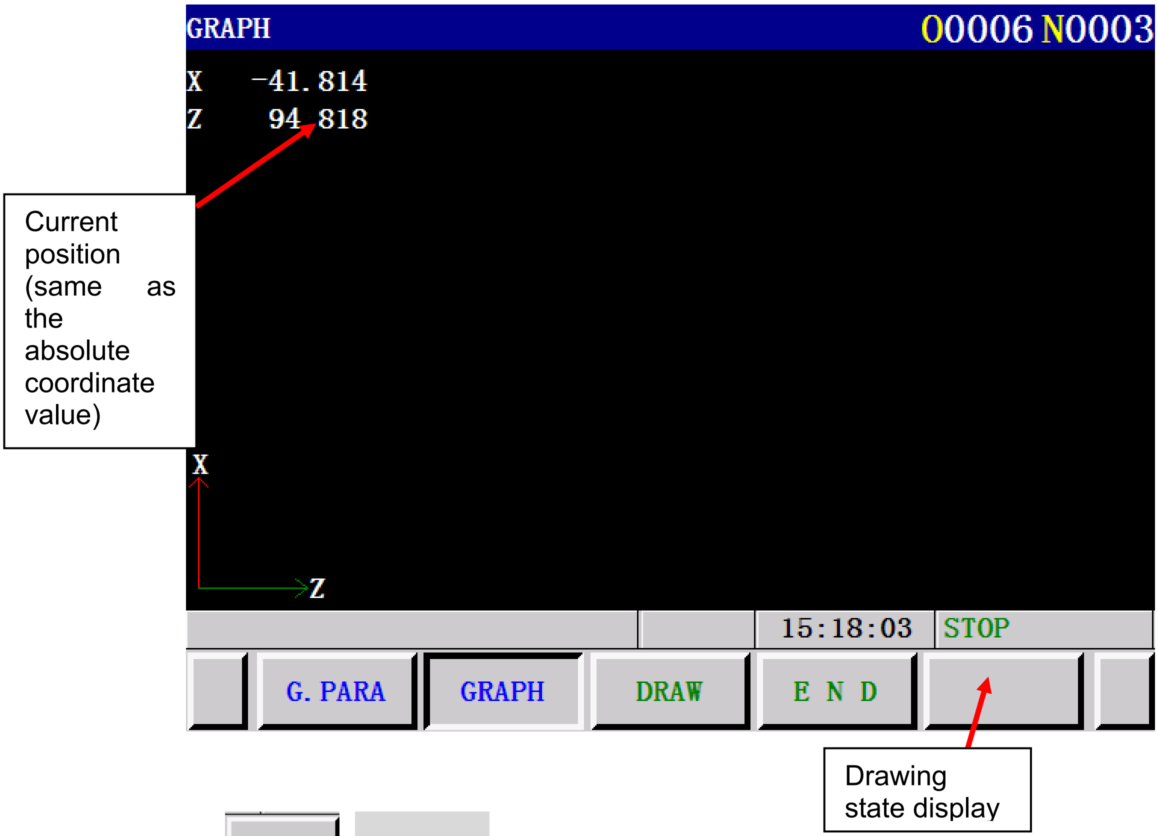
CRT CENTER X value: Set the coordinate of the drawing center in the workpiece coordinate system

CRT CENTER Z value: It will be automatically set once the range of the maximum and minimum values are set after drawing for one time.

（3）Description of the tool path

① Press 

once more when the drawing parameters are displayed, and the following figure occurs on LCD.



② Press DRAW, DRAWING is displayed on the status bar at the lower-right corner on LCD. If there has already existed the figure, it will be cleared.

③ The automatic running starts and the tool path is given.

④ Once press E N D, the drawing stops. And then press DRAW key, restart drawing after the previous figure disappears.

Note 1: The drawing is operated based on the changed coordinate value during automatic running; therefore, the program should be started during automatic running. The machine should be locked if only drawing is required.

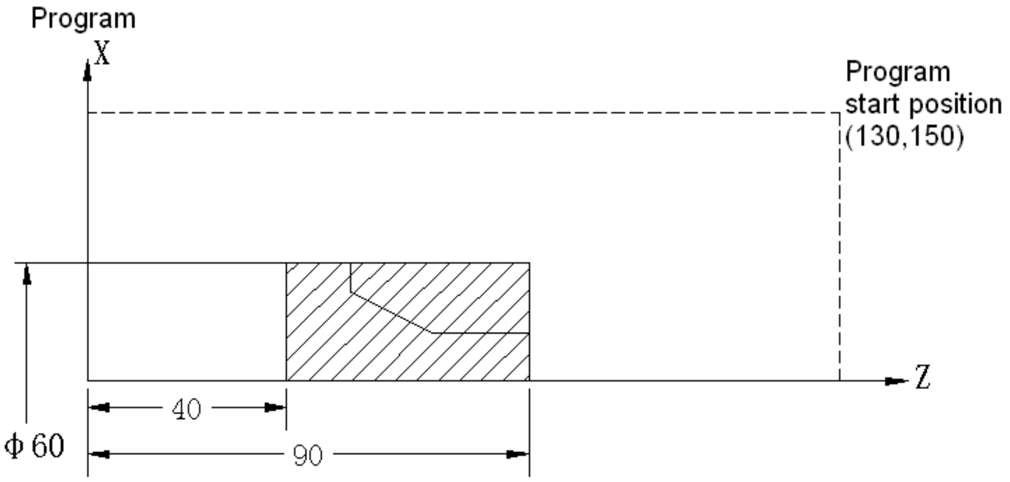
Note 2: When the feedrate is too high, the drawing can't be performed correctly. Then, decelerate in dry run mode for drawing.

（4）Example

Conditions: Specified by the diameter

Input unit 0.001mm

Drawing plane 1（AXES=1）



（a） For all tool path drawing

If the maximum and minimum values are set as below, the drawing center becomes（35，75）.

XMAX=70000，XMIN=0

ZMAX=150000，ZMIN=0

Drawing override H=144/150=0.96

Drawing override V=216/130=1.66

Drawing override 0.96（96）

（b） Drawing the shadow part

XMAX=30000，XMIN=0

ZMAX=90000，ZMIN=40000

Drawing override H=144/50=2.28

Drawing override V=216/60=3.6

Drawing override 2.88（=288）

（c） To offset the drawing position, change the maximum and minimum values with one value.

MAX+α，MIN+α

α<0， go upward, offset in the right side.

α>0， go downward, offset in the left side.

4.5 Display by the Position Displayer (Selected Function)

The position displayer shows the current position.

The position displayer is with the resetting button of the position display of each axis; after pressing the button, the displayer corresponding the axis is cleared to 0, and then, the position of the origin is shown.

The coordinate value can also be set into the position displayer through setting the parameters（PPD）, or the coordinate system (G50) is set by the program or MDI.

Note 1: When the lock switch is ON, the position displayer doesn't change; and when the switch is ON, the movement isn't displayer on the displayer, so the actual movement amount is different with the displayed value.

Note 2: The position is displayed in inch in inch input; and it is displayed in mm in mm input. During switching from

mm to inch, or from inch to mm, the position displayer is cleared to 0 after pressing the resetting button. No matter from mm to inch, or from inch to mm, the switched displayed data don't change, only the position of the decimal point will change.

Note 3: For the compensation amount, like the backlash amount, etc which belongs to the mechanical type, the displayer doesn't display.

4.6 Method of Setting Tools

A. Mechanical zero return tool setting

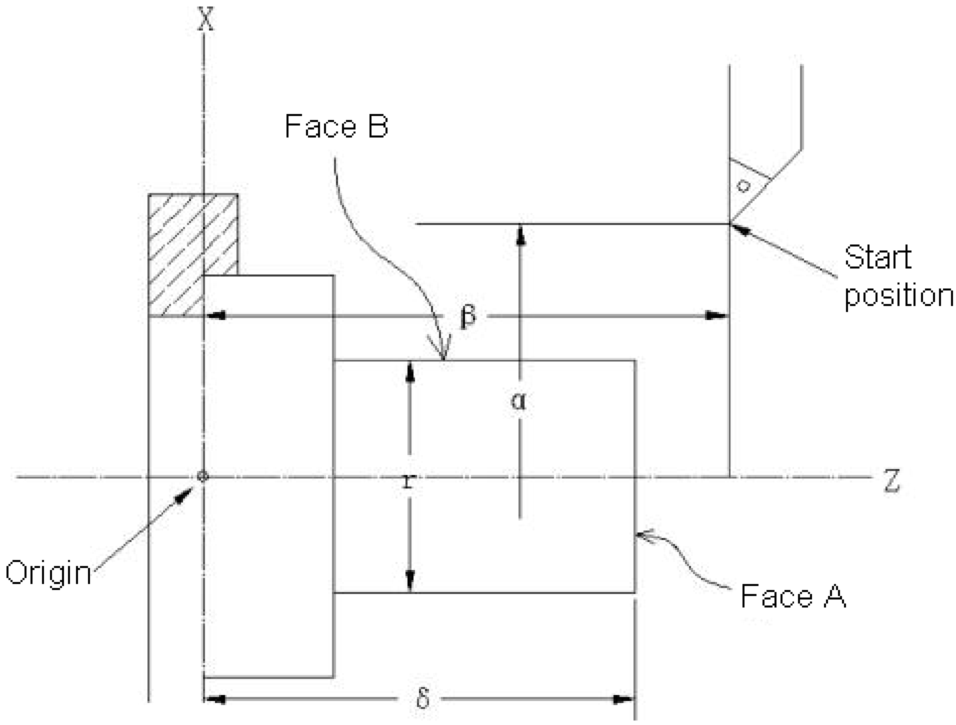
1） Press 

to enter the mechanical zero return mode, and then, X an Z axes return to the mechanical zero;

2） Select any one of tools;

3） Start the spindle, the tool is cut along the workpiece end face (it is assumed as face A), after pressing 

, the tool is retracted from face A, the spindle stops, the length of δ is measured (take point O as the workpiece zero coordinate (the workpiece zero is the origin)); refer to the following figure:



4） Press 

and 

to enter the interface of 

. It's assumed to use #01 tool offset, input "N 101"(#100+01 tool offset), and then press 

; the lower-left "N" on the screen is twinkling and then input "Z" and δ value measured previously, after

pressing 

, the system will calculate the offset value of which the tool on Z axis;

5) Start the spindle, the tool cuts along the workpiece face (it's assumed as face B), after pressing 

, the tool is retracted from face B, the spindle stops, the length of r is measured;

6) After entering the interface of "tool outline offset", #01 tool offset is continued to use, and the lower-left corner "N" on the screen is twinkling, and the cursor stops below the tool offset number of "G01" group. And, the current is the tool offset number "N101"; then, input "X" and the value of r measured previously; after pressing 

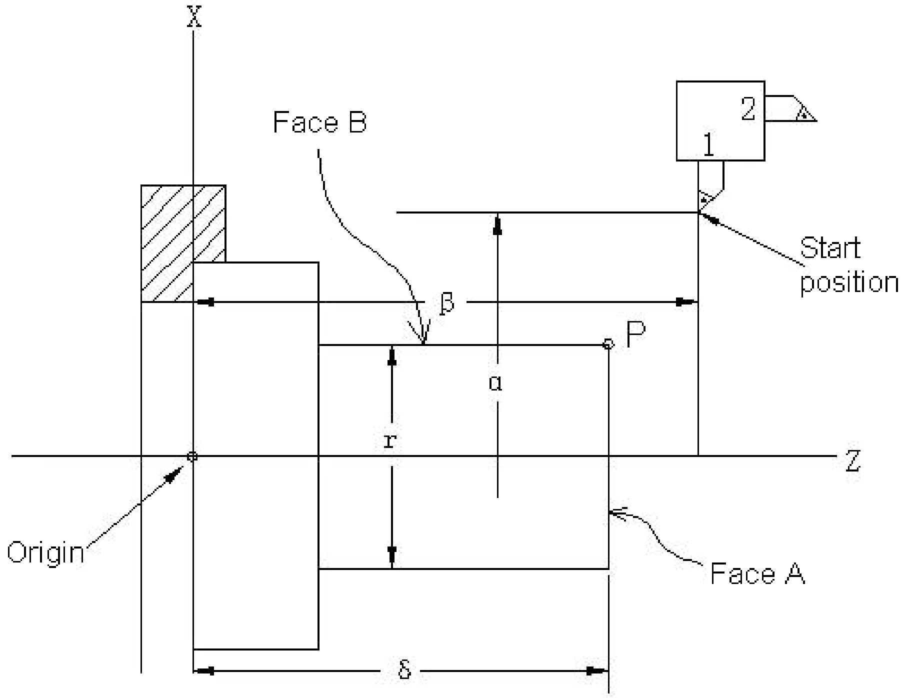
, the system automatically calculates the offset amount of which the tool on X axis; then the tool offset value is set.

Note 1: The maximum offset value of the tool in ±999.999mm in metric system or ±99.9999inch in inch system.

Note 2: Before setting the tool offset value, the tool offset compensation amount is null, that is to say, at the system initial power-on state or the executing tool offset value as 0, so it means the coordinate system isn't with the tool offset value.

B. Tool setting in the specified position

1) Select any one tool as the standard one (it's assumed that #1 tool is the standard one) , which is shown as the following figure:



2) Start the spindle;

3) After the 1st tool nose is moved to point P, in MDI mode, press 

to turn into the page of "transfer to the next block"; and in the coordinate interface, "U" of the coordinate display is twinkling on the screen after pressing **U**, and U numerical value of the relative coordinate is

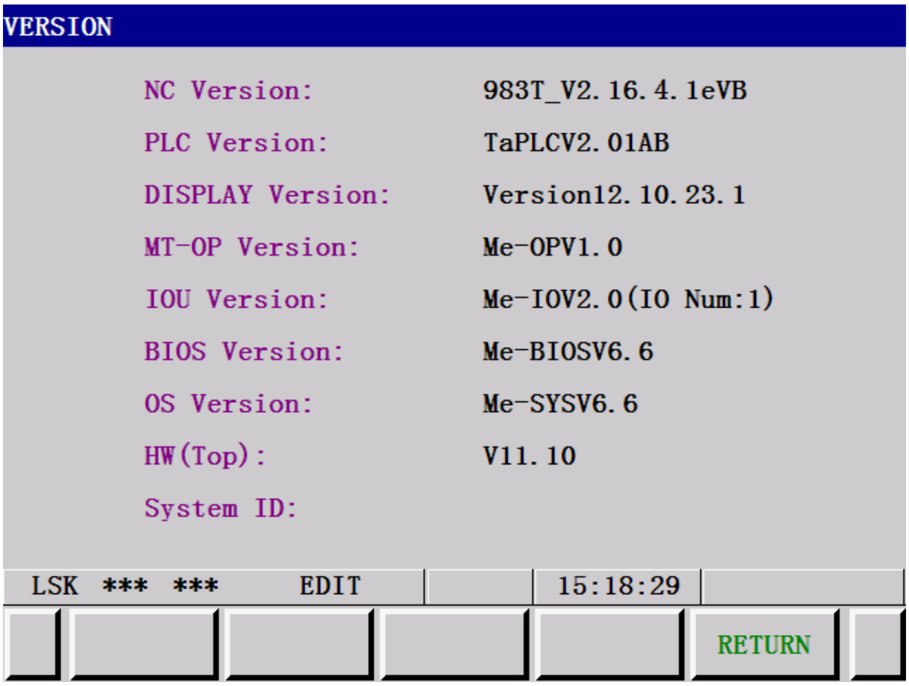
cleared with SHIFT; same, after pressing **W**, "W" is twinkling, W numerical value of the relative coordinate can also be cleared with SHIFT; then, the position of the standard tool is set, and #2 tool is changed after retracting into the safe position;

4）After #2 tool nose is moved to point P, press OFFSET and GEOM to enter the interface of GEOM. It's assumed to use #2 tool offset, press INPUT to input "N 02", the cursor stops below the tool offset number of "G02" group; then, press **U** key, "U" is twinkling on "present coordinate: (relative coordinate) , and then press INPUT. Then, the coordinate value of "U" is directly input into X tool offset value of "G02" group, and press **W**, "W" is twinkling, and then, press INPUT, the coordinate value of "W" will be directly input into Z tool offset value in "G02" group; then, the offset values of #2 tool and standard one are set.

Appendix 1 System Version Information

Check the information of the system version:

Press diagnosis key to enter the diagnosis interface, and press softkey VERSION, which is shown as the following figure.



Appendix 2 List of G codes

The following G codes can be used, and the special G codes can also be used by setting the parameters.

Standard G code	Special G code B	Special G code C	Group NO.	Function	Basic/select
*G00	*G00	*G00	01	Position (rapid)	B
*G01	*G01	*G01		Linear interpolation (cutting feed)	B
G02	G02	G02		Arc interpolation CW circle	B
G03	G03	G03		Arc interpolation CCW circle	B
G04	G04	G04	00	Dwell	B
G07	G07	G07		Feedrate sine curve control	O
G10	G10	G10		Setting the offset value	O
G20	G20	G70	06	Input in inch system	O
G21	G21	G71		Input in metric system	O
G22	G22	G22	04	Stroke limit function connected in memory type	O
G23	G23	G23		Stroke limit function disconnected in memory type	O
G27	G27	G27	00	Reference position return check	O
G28	G28	G28		Reference position return	O
G29	G29	G29		Return from the reference position	O
G30	G30	G30		The 2nd reference position return	O
G31	G31	G31		Skip function	O
G32	G33	G33	01	Linear, taper thread cutting	B
G34	G34	G34		Changeable lead thread cutting	O
G36	G36	G36	00	Automatic tool compensation X	O
G37	G37	G37		Automatic tool compensation X	O
*G40	*G40	*G40	07	Tool nose R compensation cancel	O
G41	G41	G41		Tool nose R compensation left	O
G42	G42	G42		Tool nose R compensation right	O
G50	G92	G92	00	Setting the coordinate system and the spindle maximum speed	B，O
G65	G65	G65		Just calling the user macro	O
G66	G66	G66	12	Calling the custom macro modal	O
G67	G67	G67		Cancel calling the custom macro modal	O
G68	G68	G68	13	Double tool post mirror image connected	O
G69	G69	G69		Double tool post mirror image disconnected	O
G70	G70	G72	00	Finishing cycle	O
G71	G71	G73		Outer dimension roughing cycle	O
G72	G72	G74		End face roughing cycle	O
G73	G73	G75		Closed-loop cutting cycle	O
G74	G74	G76		End face groove cutting cycle and deep hole drilling cycle	O
G75	G75	G77		Outer dimension groove cutting cycle and outer dimension cutting cycle	O
G76	G76	G78		Thread cutting cycle	O
G90	G77	G20	01	Cutting cycle A	O
G92	G78	G21		Thread cutting cycle	O
G94	G79	G24		Cutting cycle B	O
G96	G96	G96	02	Circle speed constant control	O
*G97	*G97	*G97		Cancel circle speed constant control	O
*G98	*G94	*G94	05	Feed per minute	B
*G99	*G95	*G95		Feed per revolution	B
—	*G90	*G90	03	Absolute command	B
—	*G91	*G91		Incremental command	B

B: Basic function　　　　　　　　　　O: Select function

Note 1: Several G codes can be commanded in one block, but only the last one is valid when the specified G codes are from the same group.

Note 2: G codes are modal in 01～13 groups, if soma G code is commanded, it remains valid unit the other one from the same group is specified. When CNC device is in the resetting state, the initial G codes can be restored by setting parameters, but the codes with mark * are the initial ones. In one group, if there are two G codes with * , the initial one is set by the parameter, but G20 and G21 keep the commanded states before power on.

Note 3: G codes of group 00 are non-modal and are only valid in the specified block.

Note 4: Only one G code from each group is displayed.

Note 5: If there are no commanded G codes in the list, or G codes are not correspond to the selected function, #10 alarm occurs, but G60 and G61 don't cause alarm

Appendix 3 List of the Range of Commanded Values

		Address	Input/ output in metric system	Input in inch system/Output in metric system	Input in metric system/Output in inch system	Input/ output in inch system
Min. setting unit			0.001mm	0.0001mm	0.001mm	0.0001inch
Max. stroke (distance off the reference position)			±99999.999mm	±99999.999mm	±3937.0078inch	±9999.9999inch
Max. commanded value		X，Z，U，W，I，K，	±99999.999mm	±3937.0078inch	±99999.999mm	±9999.9999inch
		R	0~99999.999mm	0~3937.0078inch	0~99999.999mm	0~99999.999mm
Cutting feedrate (override is 100%)	Feed per minute	F	1mm/min ～15000mm/min	0.01inch/min ～600.00inch/min	1 inch/min ～15000inch/min	0.01inch/min ～600.00 inch/min
	Feed per revolution		0.01mm/r ～500.00mm/r	0.0001inch/r ～50.0000inch/r	0.01mm/r ～500.00mm/r	0.01 inch/r ～50.000 inch/r
Cutting feedrate 1/10 (set by parameters)	Feed per minute		0.1～15000.0 mm/min	0.01inch/min ～600.00inch/min	0.1mm/min ～15000.0mm/min	0.01inch ～600.00inch
	Feed per revolution		0.001mm/r ～500.000mm/r	0.0001inch/r ～50.0000inch/r	0.01 mm/r ～500.000mm/r	0.0001 inch/r ～50.0000inch/r
Rapid speed (each axis independently)			60 mm/min ～15000mm/min	60 mm/min ～15000mm/min	6.0 inch/min ～600.0inch/min	6.0 inch/min ～600.0 inch/min
Cutting feedrate upper limit value Manual rapid feedrate FO			6 mm/min ～15000mm/min	6 mm/min ～15000mm/min	0.6 inch/min ～600.0inch/min	0.6 inch/min ～600.0inch/min
Manual continuous feedrate			1mm/min ～2000mm/min	0.04inch/min ～78.7inch/min	0.5 mm/min ～1016mm/min	0.02 inch/min ～40inch/min

Thread lead	F code	F	0.01mm ～500.00mm	0.0001 inch ～50.0000inch	0.01mm ～500.00mm	0.0001 inch ～50.0000inch
	E code	F	0001 mm ～500.000mm	0.000001inch ～9.999999inch	0.0001mm ～500.0000mm	0.000001 inch ～9999999 inch
Thread lead (1/10 feedrate set by parameters)	F code	F	0.001mm/r ～500.000 mm/r	0.01 inch/r ～50.0000inch/r	0.001mm/r ～500.000 mm/r	0.01 inch/r ～50.0000inch/r
	E code	E	0.00001mm ～9999999mm	0.000001 ～9.999999inch	0.00001mm ～99.99999mm	0.01 inch ～9.999999inch
The spindle max. revolution number			5000r/min	5000r/min	5000r/min	5000r/min
The coordinate value of the 2nd reference position (value off the reference position)			0mm ～ ±99999.999mm	0mm～ ±99999.99mm	0mm ～±3937.0078inch	0inch ～ ±9999.9999inch
Tool offset amount		OFX, OFZ, OFR	0mm ～±999.999mm	0inch ～±99.9999inch	0mm ～±999.999mm	0inch ～±99.9999inch
Min. value of incremental feeding			0.001mm	0.0001inch	0.001mm	0.0001inch
Backlash compensation amount			0 mm～0.255mm	0 mm ～ 0.255 mm	0～0.025inch	0～0.0255inch
The storable or the stroke limit range (distance off the reference position)			±99999.9mm	±99999.999mm	±3937.0078 inch	±9999.9999inch
Dwell		X，U	0 s～99999.999s	0 s ～ 99999.999s	0 s～99999.999s	0 s ～ 99999.999s
		P	0ms ～99999999ms	0m s ～99999999ms	0ms ～99999999ms	0ms ～99999999ms
Program number		：（ＩＳＯ） O（EIA）	1～9999	1～9999	1～9999	1～9999
Sequence number		N	1～9999	1～9999	1～9999	1～9999
Preparatory function		G	0～99	0～99	0～99	0～99
Spindle function		S	0～9999	0～9999	0～9999	0～9999
Tool function		T	0～9932	0～9932	0～9932	0～9932
Miscellaneous function		M	0～99	0～99	0～99	0～99
Specified sequence number		P，Q	1～9999	1～9999	1～9999	1～9999
Repeated times		L	1～9999	1～9999	1～9999	1～9999
Para meters	Angle	A	Specified value			
	Cutting amount	D，I，K	Same coordinate word			
	Times	D	1～9999	1～9999	1～9999	1～9999

Appendix 4　The Calculated Diagram

D4.1　Incorrect Thread Length

The incorrect parts δ1 and δ2 of the thread will occur during the automatic acceleration and deceleration and cutting thread, which is shown as the figure 4.1. Therefore, the surplus of δ1 and δ2 should be reserved.

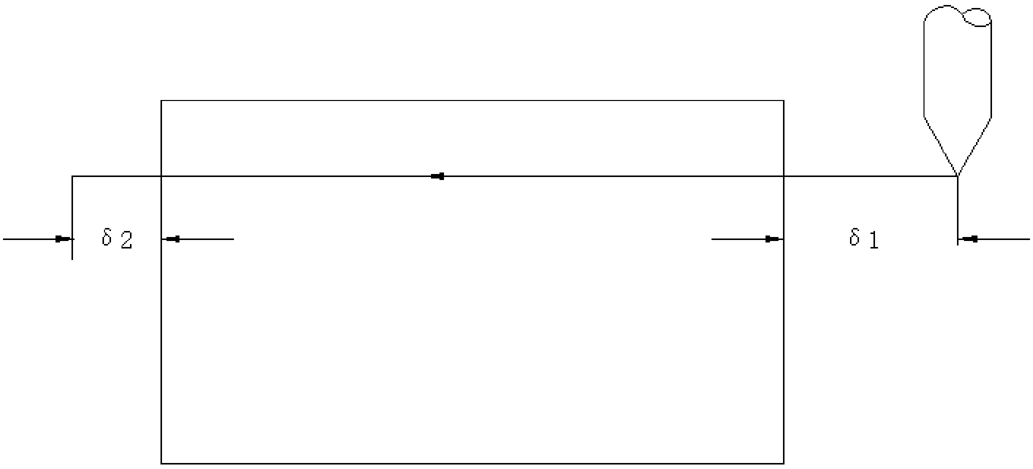


Fig. 4.1 Incorrect thread path

（1）　δ2 is set by the thread cutting speed V（mm/sec）and the servo time constant T1.

δ2=T1·V　（mm）………………（1）

Unit: T1：sec,V:mm/sec

V is set by the thread lead L and the spindle rotation revolution R.

$$V=\frac{1}{60}\cdot R\,(\text{rpm})\cdot L(\text{mm})$$

O the servo time constant T1（sec）is always calculated by T1=0.033.

（2）　Method of setting δ1

δ1 is set by the thread cutting speed V, the servo time constant T1 and the thread precision a.

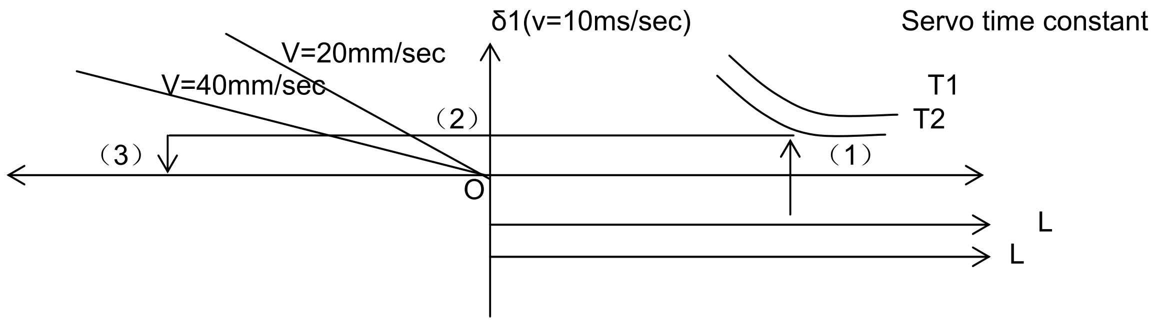
$$\delta 1=\{t-T1+T1\exp(-\frac{t}{T1})\}\ V\ \text{………………}\ (2)$$

$$a = \exp\ (-\frac{t}{T1})\ \text{………………}\ (3)$$

When the thread cutting begins, the specified lead is less than the set lead L, so the lead error is allowed. Therefore, $a = \frac{L}{L}$

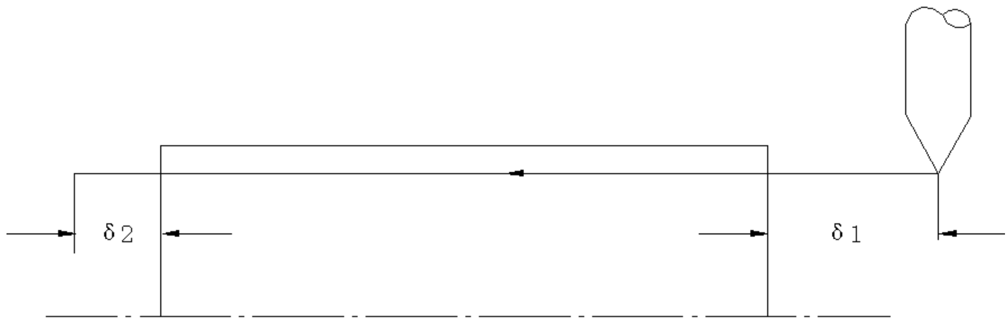
When the value of a is set, the time t of reaching the required precision is set by formula （3）and value t is put into formula (2) to obtain δ1; moreover, V，T1and T2 are set with the same method of δ2.

Because the calculation of δ1 is very difficult, so the figure of δ1 is shown as below, refer to the following introduction.



Initially, the thread precision a is obtained by known thread level and the thread value in position (1) , and δ1 is obtained when V=100mm/sec based on the acceleration and deceleration time constant during the mechanical cutting; then, δ1 of the other speed can be obtained based on the thread cutting speed in position (3).

D4.2 Simple Calculation Method of the Incorrect Thread Length



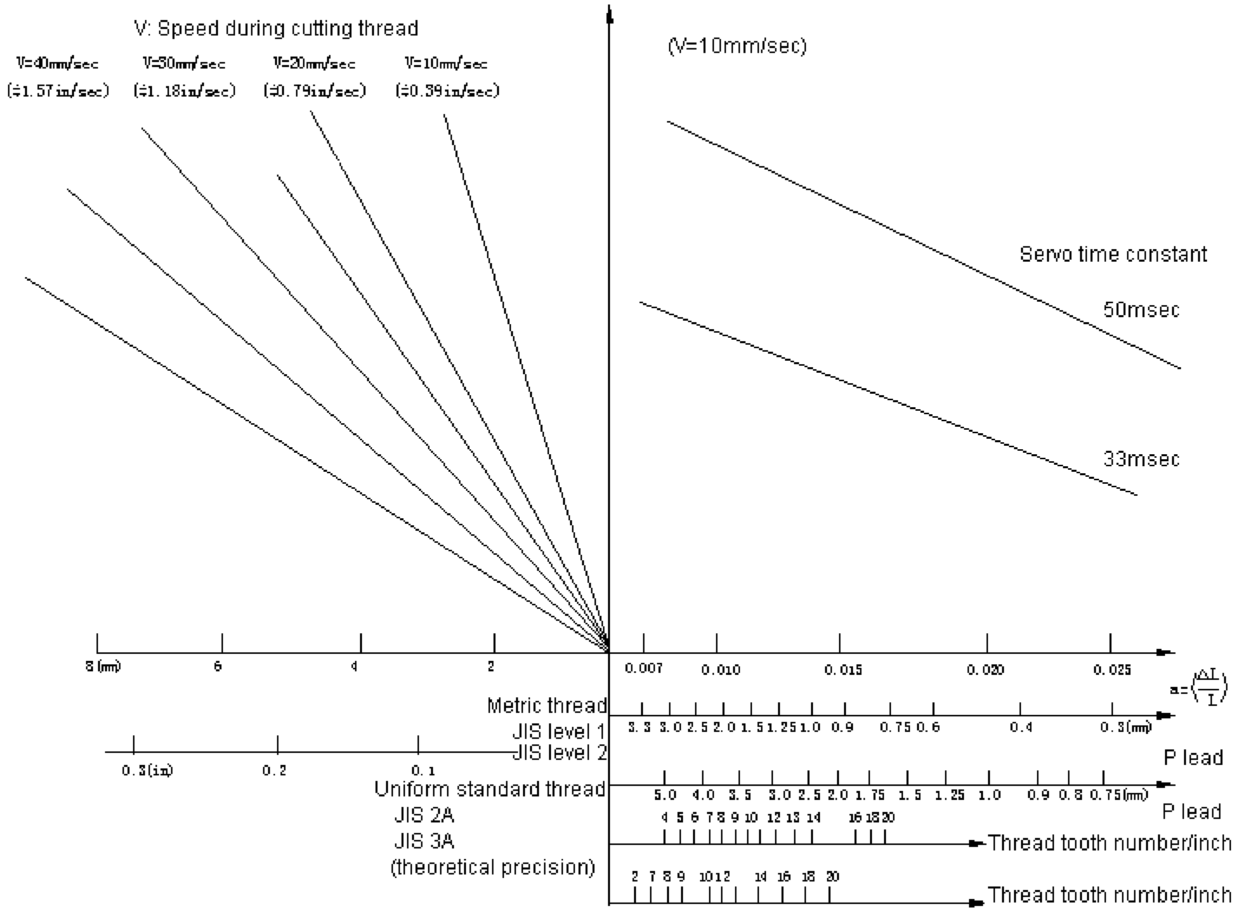
(1) $\delta 2=\dfrac{L \cdot R}{1800*}$ (mm)

L: thread lead（mm） R: number of the spindle revolutions（rpm）

(2) $1=\dfrac{L \cdot R}{1800*}\cdot(-1-Lna)$

$=\delta 2\cdot(-1-Lna)$

* When the servo time constant T=0.033,



Calculation diagram for obtaining the closest distance δ1

As follows, a is the thread allowance error, and "－1－Lna" is calculated:

a	－1－Lna
0.0005	4.298
0.01	3.605
0.015	3.200
0.02	2.912

(Example)　R=350rpm

L=1mm

when a =0.01

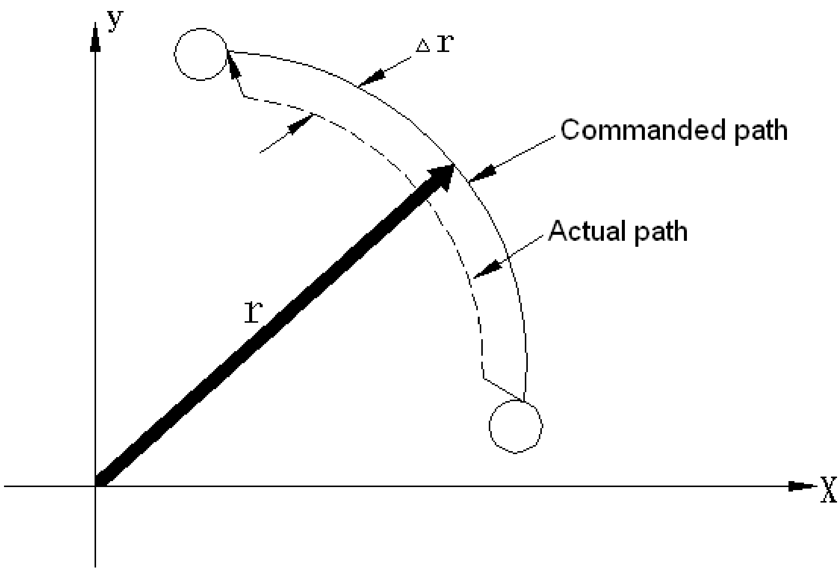
$$\delta_2=\frac{350\times1}{1800}=0.194\text{mm}$$

$$\delta_1=\delta_2\times3.605=0.701\text{mm}$$

D4.3　Errors during Arc Cutting in the Radius Direction

When the servo motor is used, the errors occurs between the input and the output axes because the motor lags behind. During the linear interpolation, the error doesn't exist during the tool is moved on the commanded path; while in arc interpolation, the error occurs especially in the arc cutting at high speed in radius direction. And the error can be obtained with the following process.

Appendix 4　The Calculated Diagram



△r: Radius error maximum value（mm）

V: Feedrate（mm）/sec

r: Arc radius

T1: The index acceleration and deceleration time constant during cutting （sec）（T1＝0）

T2: Time constant of the motor（sec）

(Note) The index acceleration and deceleration time constant is 0 in the system.

$$\triangle r=\frac{1}{2}\left(T_1^2+T_2^2\right)\bullet\frac{V^2}{r} \qquad \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots(1)$$

In the actual machining, the machining radius r（mm）and allowance error △r（mm）of the workpiece are given, so the allowed speed V（mm/sec）can be obtained with formula (1).

The acceleration and deceleration time constant during cutting vary with the different machines, please refer to the manual from the machine manufacturer.

The time constant of the servo motor is calculated with $T_2=33\times10^{-3}$（sec）.

Appendix 5 Parameters

When CNC is connected with the servo motor or CNC is connected with the machine, the parameters should be set to play full role of the characteristics of the servo motor and the machine. And the parameters are relative with the machine, please refer to the parameter list provided by the machine manufacturer.

E5.1 Display of Parameters

（a）Press .

（b）Two method of looking up the parameters

Method A:

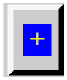
Press page up key, the interface is paged up.

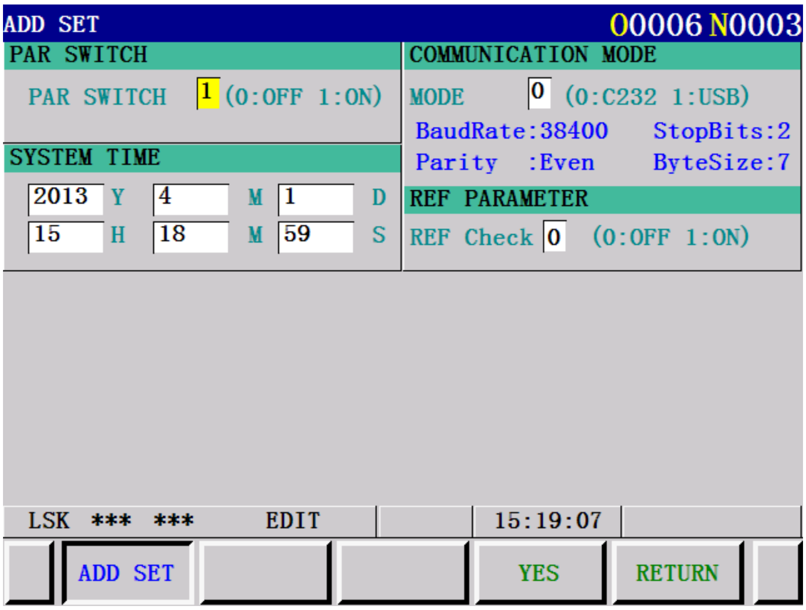
Press page down key, the interface is paged down.

Method B:

Press N + parameter number to be looked into + input to look up the corresponding parameters.

E5.2 Setting Parameters

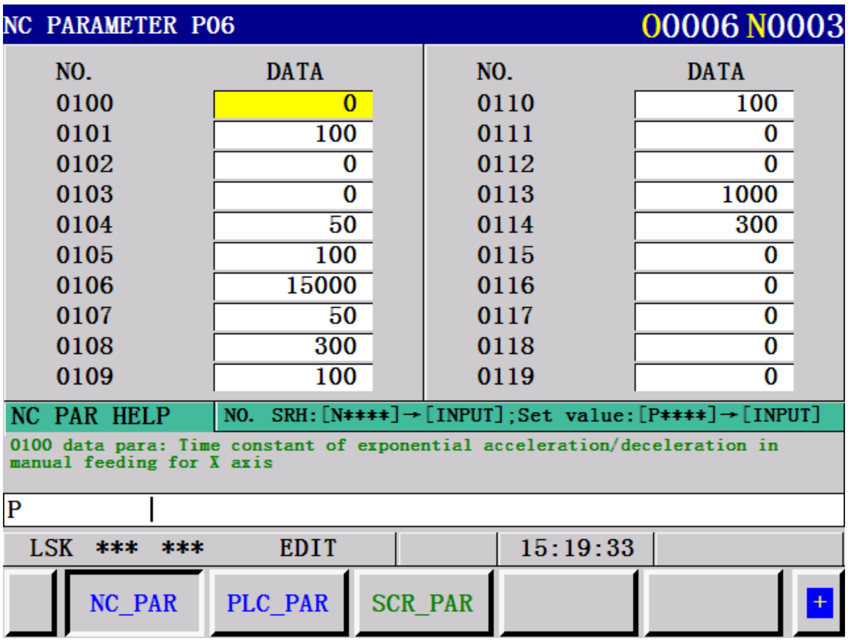
（a） Press  on the display panel to enter parameter interface, and then press the extension key  on the small keypad below the screen, and then press the softkey  to enter into the miscellaneous setting interface, and set the parameter switch as 1, press enter button, it prompts the setting is done. The following figure shows the parameter switch is ON and the parameter switch state still remains the one after power off:



（b）Select MDI mode or in the emergency stop state.

(c) Press  to enter the parameter interface.

(d) Press [N] + [parameter number to be set] + [input]. The interface to set the parameter number is selected and the base color of the parameter data is changed into yellow. (with the page keys and the cursor keys [↓] [↑] can also make it), which is shown as the following figure:



(e) Set with [P] + [setting data] + [input]

Press [cancel] key when the input is wrong.

(f) Confirm whether the setting is correct.

(g) After all the parameters are set and confirmed, the parameter switch is OFF.

(h) Press reset button to clear the alarm state（NO． 100）. If it is in the emergency stop state, release it.

E5.3 List of Parameters

List of parameter numbers.

Parameter number	Content
0000—0004	Fixed parameters
0005—0010	Various parameters
0012	Reference position return method and direction
0013	Gear ratio between the spindle and the position encoder
0014—0015	DMR, refer to the capacity of the counter
0018	The frequency of the backlash, and the input unit is 1/10.
0019	Time width of MF, SF, TF, BF and FIN
0020	Whether the reference position return function is valid
0024—0026	Various parameters
0027—0028	CMR
0031—0032	Minimum limited value of VLOC
0035—0036	M codes of non-buffer
0037	Spindle motor rotary speed when the spindle gear is changed
0038	Spindle rotate speed during the spindle exact stop
0040	The allowance value of the limit check in the external tool compensation

	II—B function
0041—0056	Data override of the outer dimension measurement of the external tool nose compensation II—B function
0057—0059	Machining time
0062	Spindle speed arrival signal timer
0064	Chamfering width during the thread cutting
0065	Retraction amount (return amount) of G74 and G75 modes
0066	Finishing surplus in G76 mode
0067	Retraction amount in G71 and G72 modes
0068	Minimum cutting depth of the thread cutting in G76 mode
0069	During the tool compensation, the very small movement amount in the machining situation of the acute angle closed to 90º is ignored.
0070—0071	In-position width
0074—0075	The position offset limit value during stop.
0078—0079	The position offset limit value during moving
0082—0083	Grid offset amount
0086—0087	Servo loop gain override
0090	Servo loop gain
0091	JOG feedrate
0092—0093	Rapid movement speed
0096—0097	Linear acceleration/deceleration time constant (for the rapid traverse rate)
0100—0101	Manual feeding time constant
0104	X axis acceleration/deceleration time constant in the thread cutting mode
0105	Cutting feed acceleration/deceleration time constant
0106	Upper feedrate during the cutting feed
0107	Lower limit value of X axis in the thread cutting mode
0108	Lower limit value of acceleration/deceleration in the thread cutting mode
0109—0110	Lower limit feedrate of manual feeding in the acceleration/deceleration
0113	Minimum speed (F0) during the rapid movement override
0114	Lower speed during the reference position return
0115—0116	Backlash compensation amount
0119	The spindle offset compensation amount (S analog output)
0120—0123	When the spindle speed command is 10 V, the spindle speed are corresponded to the gears 1~4.
0124-0125	Offset compensation amount
0132	The spindle minimum revolution speed (G96) in the constant surface speed control mode.
0133	Measuring speed during the automatic tool compensation
0140	The spindle speed gain adjustment (analog output in S4 bits)
0141—0142	Setting the operation time
0143—0158	Setting the stroke limit in the memory type
0159—0160	The 2nd reference position
0163—0164	Value r of X and Z axes in the automatic tool compensation
0165—0166	Value ε of X and Z axes in the automatic tool compensation
0300—0304	Fixed parameters
0305—0308	Various parameters
0310—0313	Baud rate of I/O device
0316	Parameters about the rotary transformer and the inductosyn
0317	The codes corresponding to “#” (custom macro)
0319	Parameters about the macro
0320—0322	Macro calling by M codes
0323—0332	Macro calling by G codes
0340	Select the input device
0341	Select the output device

0342	Skip cutting speed
0343—0346	The spindle minimum speed controlled in the constant surface speed corresponding to the gear 1,2,3,4.
0358	Maximum input value of the incremental wearing offset
0375—0376	Setting value of the automatic coordinate system input in mm (metric system)
0379—0380	Setting value of the automatic coordinate system input in inch (inch system)
0388	Maximum value of the wearing offset

List of parameter function

Function	Para. NO.	Content
Servo	005	Whether the servo disconnect signal is valid or invalid
	006	·Whether the automatic drifting compensation is valid or not ·Whether execute the in-position check
	014—015	Detect the override setting(DMR)
	026	Whether the alarm occurs when VRDY signal is connected before output PRDY
	027—028	Setting command override (CMR)
	031—032	Setting the feeding commanded limited value
	070—071	In-position width
	074—075	Limited value of the position offset amount during stop
	078—079	Limited value of the position offset amount during moving
	082—083	Grid offset amount
	086—087	Servo loop gain override
	090	Servo loop gain
	124—125	Drift compensation amount of X and Z axes
	128—129	Servo phase displacement amount
	316	Whether execute the CDSCG feedback frequency detection
	316	DSCG position detection system is set for the rotary transformer, the inductosyn or the pulse encoder
Spindle function	005	Output in S4 bit or S code (BCD)
	009	S4 bit code output the voltage symbol (analog output)
	009	Setting the spindle override function (SOV)
	010	Select the spindle stop signal for the normal open contact or NC contact (*SSTP)
	037	The number of the spindle motor revolution during the spindle gear change
	038	The number of the spindle revolution during the spindle exact stop
	062	Setting the delay time of the spindle speed arrival signal
	119	The spindle speed offset compensation value (S4 bit analog output A/B)
	120—123	When the spindle speed commanded voltage is 10V, the spindle maximum revolution number corresponding to the gears 1～4.
	132	The minimum spindle revolution number in the constant surface speed control
	134	The allowed maximum spindle speed
	140	S4 bit analog output gain adjustment
	307	S4 bit binary system 12 bits output/ analog output low speed limited value, and it is valid for all axes or the single axis.
	0343—0346	The spindle minimum speed in the constant surface speed control corresponding to the gears 1,2,3,4.
Reference posit	010	When the reference position return decelerates, set the acceleration as 1 or 0(*DECX, *DECZ).
	012	Method and direction of the reference position return

	014—015	Capacity of the reference counter of each axis
	020	Whether the reference position return function is valid or not
	082—083	Set the grid offset amount of each axis
	114	Set the low speed rate after the reference position return decelerates
	159—160	Distance of each axis from the 2nd reference position to the 1st one
Tool compensation	007	Set the incremental value or the absolute value during the offset amount input
	008	Specified by the radius or the diameter during the offset amount input
	010	Whether the data can be set when the program lock is OFF on the machine side
	040	The limit detection allowance value of the external tool compensation B (forbid to use)
	040—056	The data override of the outer dimension of the external tool compensation B (forbid to use)
	069	Set the very small movement amount is ignored during the cutter compensation
	133	Set the measuring speed during the automatic tool compensation
	163—164	Set r in X and Z axes during the automatic tool compensation
	165—166	Set ε in X and Z axes during the automatic tool compensation
	0358	Maximum input value of the incremental wearing offset
	0388	Maximum value of the wearing offset
Backlash compensation	012	After power on, the initial direction of the backlash
	018	Backlash compensation pulse frequency (256KHz)
	115—116	Backlash compensation amount of X and Z axes
Canned cycle	008	Whether the dry run is valid during the thread cutting
	009	Whether execute the canned cycle in the block without the movement commands
	064	Set the chamfering width of the thread cutting (G92, G76)
	065	Retraction amount in G74 and G75 modes
	066	Finishing amount in G76 mode
	067	The retraction amount of multiple cycles in G71, G72 modes
	068	Set the minimum cutting amount in thread cutting cycle G76 mode
	307	Whether execute the finishing of the last outline cutting in G71 and G72 modes
Macro command function	025	Macro cutting parameters
	053	M codes making the macro interruption valid
	054	M codes making the macro interruption invalid
	306	Whether the macro can be called by T codes
	308	Whether read and write DI, DO through macro variable
	314	The macro is interrupted in the middle or at the end of the block
	317	The parameters stored in the macro same as the codes corresponding to "#"
	318	Set the various parameters for the macro
	319	Same as above
	320—322	Setting M codes of three types for calling the macro
	323—332	Setting G codes of ten types for calling the macro

E5.4 Precautions of Using the Parameters

1) The parameter contents of #0000-0004 and 0300-0304 vary as the different machines, please refer to the parameter list attached with NC.

2) Please set as "0" for the parameters of which purpose are not introduced.

3) The contents of the parameter number with the format | | | | | | | | | are set

Appendix 5 Parameters

by the binary system (1 or 0).

The contents of the parameter number with the format are set by the decimal system.

4）Don't set the sign of "-", because it doesn't exist in the parameters.

5）The forbidden parameters must be set as "0".

6）The range is set while the parameters are not used, they can be set as "0".

E5.5 Detailed Introduction of Each Parameter

Para. NO.	Bit 7	Bit 6	Bit 5	Bit 4	Bit 3	Bit 2	Bit 1	Bit 0
0005	ORWD	EIT	EENB	SCD	FMIC	MDL	MIC	SCW

Detailed introduction		
	Set as 1	Set as 0
ORWD	Unused	
EIT	The interlock signal is valid	The interlock signal is invalid
EENB	Servo OFF signal is valid	Servo OFF signal is invalid
SCD	Even S4 bit (analog output) or S4 bit (12 bits output in binary system) (any function) is selected, S4 bit (analog) or S4 bit (12 bits in binary system) command isn't sent, while S code (BCD) is output.	If S bit (analog output) or S4 bit (12 bits output in binary system) (any function) is selected, the corresponding S4 bit (12 bits output in binary system) (any function) is selected, and the corresponding S4 bit code is output; otherwise, S code (BCD) is output.
FMIC	In the metric input, the unit of the feedrate is 1/10.	Not 1/10.
MDL	On the position displayer, in the metric input, the displayed minimum unit is 0.01mm; in the inch input, it is 0.0001inch and remains unchanged.	On the position displayer, in the metric input, the displayed minimum unit is 0.01mm; in the inch input, it is 0.0001inch and remains unchanged.
MIC	In the metric input, the minimum set unit is 0.01mm; in the inch input, it is 0.0001inch and remains unchanged.	In the metric input, the minimum set unit is 0.01mm; in the inch input, it is 0.0001inch and remains unchanged.
SCW	The minimum movement unit is 0.0001inch (the machine in inch system).	The minimum movement unit is 0.001mm (the machine in metric system).

Para. NO.	Bit 7	Bit 6	Bit 5	Bit 4	Bit 3	Bit 2	Bit 1	Bit 0
0006	ADFT	EOM	CINP	DCS	CLER	TVC	PPD	RDRN

Detailed introduction		
	Set as 1	Set as 0
ADFT	Execute automatic drift compensation	Not execute automatic drift compensation
EOM	When M30 is commanded, M30 is sent to the machine side and FIN signal is returned, then, the block is continuously executed from the head of the program; or FIN signal doesn't return from the machine side while the external resetting signal is returned, then the program is returned to the head and the system becomes the resetting state (in Auto mode).	After commanding M30, only M30 is sent from the machine side, and the reset & rewind signals are not used, the program doesn't return to the head (in the memory mode).
CINP	Between two non-cutting blocks, or the error detection is valid, the commanded speed decelerates into 0 and confirm the mechanical position has already reached the commanded one, then the next block can be executed (the confirmation is called as the in-position detection).	Between two non-cutting blocks, or the error detection is valid, once the commanded speed decelerates into 0, the next block is executed (the in-position detection is not performed).
DCS	The start button on the keypad panel is directly started from NC side not via the machine side (only in MDI mode).	The start button on the keypad panel is sent to the machine side, NC starts after it receives the start signal return from the machine side.

CLER	NC becomes the clear state with the reset button, the external reset signal, the reset and the rewind signals. (refer to Appendix 7).	NC becomes the reset state with the reset button, the external reset signal, the reset and the rewind signals. (refer to Appendix 7).
TVC	TV check is not performed in the control output part (the note part) .	TV check is performed in the control output part (the note part) .
PPD	The relative coordinate display can be preset with command G50(G92).	The relative coordinate display can be preset with command G50(G92).
RDRN	Dry run is also valid for the rapid feed command.	Dry run is invalid for the rapid feed command.

Para. NO.	Bit 7	Bit 6	Bit 5	Bit 4	Bit 3	Bit 2	Bit 1	Bit 0
0007	ICR	IOF	GSP	SCTO	G90	G98		G00

Detailed introduction		
	Set as 1	Set as 0
ICR	The ISO codes are punching output, and EOB codes are output with LF.	The ISO codes are punching output, and EOB codes are output with LF CR CR.
IOF	When the incremental offset function is selected, the input offset value is taken as the incremental one (only valid for the tool nose offset amount R).	The input offset value is taken as the absolute one (only valid for the tool nose offset amount R).
GSP	Use the special G code B.	Use the standard G codes.
SCTO	From S command and from the rapid change into the cutting feed, the speed arrival signal is detected	The speed arrival signal is not detected
G90	At power on and in the clear state, the initial state is G90 one (the special G code).	At power on and in the clear state, the initial state is G91 one.
G98	At power on and in the clear state, it becomes G98 state (the special G code is G94 in B/C).	At power on and in the clear state, it becomes G99 state (the special G code is G95 in B/C).
G00	At power on and in the clear state, it becomes G00 state.	At power on and in the clear state, it becomes G01 state.

Para. NO.	Bit 7	Bit 6	Bit 5	Bit 4	Bit 3	Bit 2	Bit 1	Bit 0
0008	ORC	RWL		TLCC	T2D	TOC	NMTN	TDRN

Detailed introduction		
	Set as 1	Set as 0
ORC	Command the offset value with the radius.	Command the offset value with the diameter.
RWL	The area outside the stored stroke limit 3 is taken as the forbidden one.	The area inside the stored stroke limit 3 is taken as the forbidden one.
TLCC	After changing the offset amount, the new one becomes valid from the next block.	After changing the offset amount, the new one becomes valid from the next T code.
T2D	The lower two bits of T code (2+2 bit) is the offset amount.	The lower one bit of T code (1+1 bit) is the offset amount.
TOC	The offset is cancelled in the resetting state.	The offset is not cancelled in the resetting state.
NMTN	When STLK signal is connected, the traverse command is not executed, while M, S and T commands are executed (STLK: interlock signal)	When STLK signal is connected, the traverse command and M, S and T commands are executed (STLK: interlock signal)
TDRN	For the thread cutting , the dry run is also valid.	For the thread cutting , the dry run is not valid.

Para. NO.	Bit 7	Bit 6	Bit 5	Bit 4	Bit 3	Bit 2	Bit 1	Bit 0
0009	TCW	CWM	SOV	SSCR	MOR	SOVC	REDT	ISOT

Detailed introduction		
	Set as 1	Set as 0
TCW	Symbols when S4 bit digit analog is output	

CWM				

TCW	CWM	CODE
0	0	M03 and M04 all are positive
0	1	M03 and M04 all are negative
1	0	M03 is positive, M04 is negative
1	1	M03 is negative, M04 is positive.

SOV	The spindle override is valid.	The spindle override is not valid.
SSCR	In the constant speed control mode, the circular speed in the rapid block is calculated with the end position of the coordinate value in X axis.	In the constant speed control mode, the circular speed of the rapid block is calculated with the current value in X axis, which is same as the cutting feed.
MOR	In the single canned cycle mode, the canned cycle is not executed even the block without the movement command is specified.	In the single canned cycle mode, the single canned cycle is not executed in the block without the movement command
SOVC	During the thread cutting（G32, G76, G92）, the spindle override is clamped in 100%.	Even in the thread cutting, the spindle override is not clamped in 100%.
REDT	In Edit mode, the part program can be stored into the memory with the cycle start button.	It can't be stored with the cycle start button.
ISOT	With the stroke limit selection in the memory type, the manual rapid feeding is also valid even the reference position return is not executed.	With the stroke limit selection in the memory type, the manual rapid feeding can be valid only after the reference position return is executed.

Para. NO.	Bit 7	Bit 6	Bit 5	Bit 4	Bit 3	Bit 2	Bit 1	Bit 0
0010	DGNE	SETE	DECI	SSPB	NPRD	PROD	CTHD	

Detailed introduction		
	Set as 1	Set as 0
DGNE	In the diagnosis state, the data output is valid.	In the diagnosis state, the data output is invalid.
SETE	The input can be set when the lock key is closed.	The input can not be set when the lock key is closed.
DECI	During the reference position return, it decelerates when the deceleration signal *DECX or *DECZ is "1".	During the reference position return, it decelerates when the deceleration signal *DECX or *DECZ is "0".
SSPB	The spindle stops when the spindle stop input signal *SSTP is "0".	The spindle stops when the spindle stop input signal *SSTP is "1". (the standard one is 0)
NPRD	The decimal point is not used for input and display.	The decimal point is used for input and display.
PROD	When the current value is displayed, display the programmed position.	Display the actual position after the tool position compensation and the tool cutter compensation.
CTHD	Continuous thread cutting (not recommended).	Not operate continuous thread cutting.

Para. NO.	Bit 7	Bit 6	Bit 5	Bit 4	Bit 3	Bit 2	Bit 1	Bit 0
0012			ZGMZ	ZGMX			ZMZ	ZMX

Detailed introduction		
	Set as 1	Set as 0
ZGMZ	Z axis reference position return mode is the magnet switch mode.	Z axis reference position return mode is the grid mode.
ZGMX	X axis reference position return mode is the magnet switch mode.	X axis reference position return mode is the grid mode.
ZMZ	Z axis reference position return direction and the gap initial position at power on is negative.	Z axis reference position return direction and the gap initial position at power on is positive.

ZMX	X axis reference position return direction and the initial position of the gap at power on is negative.	X axis reference position return direction and the initial position of the gap at power on is positive.
Note 1: For the axis with the reference position return function, the reference position return direction is same as the gap initial direction at power on; while for the axis without the reference position return function, the parameter only includes the gap initial direction. **Note 2:** After the power on, the gap compensation is executed when the movement direction is set by the parameter.		

Para. NO.	Bit 7	Bit 6	Bit 5	Bit 4	Bit 3	Bit 2	Bit 1	Bit 0
0013	PSG2	PSG1	RVZRN					

Detailed introduction		
	Set as 1	Set as 0
PSG2 PSG1	The gear ratio between the spindle and the position encoder	
RVZRN	When the reference position is returned in JOG mode, the reference position return can't be executed in the direction opposite with the return one.	The reference position return is valid in the direction opposite with the return one.

Override	×1	×2	×4	×8
PSG2	0	0	1	1
PSG1	0	1	0	1

$$\text{Override}=\frac{\text{The number of the spindle revolution}}{\text{The number of the position encoder revolution}}$$

Para. NO.	Bit 7	Bit 6	Bit 5	Bit 4	Bit 3	Bit 2	Bit 1	Bit 0
0014		DMRX			GRDX			
0015		DMRZ			GRDZ			

Detailed introduction		
	Set as 1.	Set as 0.
DMRX DMRZ	The detection override ratio is used by X and Z axes in sequence.	

Set code			Override Pulse encoder
0	0	0	1/2
0	0	1	1
0	1	0	1
0	1	1	2
1	0	0	3/2
1	0	1	3
1	1	0	2
1	1	1	4

GRDX / GRDZ: The capacity of the reference counter of X and Z axes in sequence.
Reference counter capacity = detection override * the pulse number of one revolution of the motor detector

Set code				Capacity of one cycle
0	0	0	1	2000
0	0	1	0	3000
0	0	1	1	4000
0	1	0	0	5000
0	1	0	1	6000
0	1	1	1	8000
1	0	0	1	10000

Note: If the value not listed in the above table is set, the capacity is set as 8000.

Note: DMR and GRD are set in binary system.

Method of calculating CMR and DMR

① Step one: Set the feedback pulse number of CMR and DMR

$$\frac{\text{Min. movement unit (mm/inch)}}{CMR}=\text{Detection unit}=\frac{\text{L(mm/inch)}}{DMR*\alpha*\beta}$$

Formula 1: Relation between CMR and DMR

Pulse command=Minimum movement unit: 0.001mm(metric unit)

0.0001inch (inch unit)

CMR:Commanded override ratio, the parameter is 27,28, and the set value normally is 1, and it is corresponded to the set code 2.

DMR: Detection override ratio, the parameter is 14,15.

L: Movement amount of per revolution feeding of the detector.

α:Pulse number per revolution sent from the detector of the motor.

β:The feedback pulse ratio is always 1 in GSK983Ta.

② Step two: Set the capacity of the reference counter

The reference counter is relative with the zero return in grid mode (Z phase signal), and the machine zero return stop position is not correct if the setting is not correct.

$$\text{The reference counter capacity} = \text{the movement amount per revolution of the pulse encoder/the detection unit}= DMR*\ \alpha*\beta$$

α: Pulse number per revolution sent from the detector of the motor.

β:The feedback pulse ratio is always 1 in GSK983Ta.

The list of the normal thread screw pitch in GSK983Ta

GSK983Ta X axis parameters

Movement amount per revolution of the motor			5000ppr/rev motor					2500ppr/rev motor			
Metric system	Rotary axis (degree)	(inch) Inch system	983Ta X axis parameters		GD2000 Drive electronical gear ratio			983Ta X axis parameter		GD2000 drive electronical gear ratio	
			27	14	PA41	PA42		27	14	PA41	PA42

(mm)											
1	1	0.1	2	00000001	4	5		2	00010001	4	5
2	2	0.2	2	00010011	4	5		1	00010001	4	5
3	3	0.3	2	01000101	4	5		1	01000010	4	5
4	4	0.4	2	01100111	4	5		2	01110111	4	5
5	5	0.5	2	01101001	1	1		2	01111001	1	1
6	6	0.6	2	01010101	4	5		1	01010101	4	5
8	8	0.8	1	01100111	4	5		1	01110111	4	5
10	10	1.0	1	01101001	1	1		1	01111001	1	1

GSK983Ta Z axis parameters

Movement amount per revolution of the motor			5000ppr/rev motor					2500ppr/rev motor			
(mm) Metric system	Rotary axis (degree)	Inch system (inch)	983Ta Z axis parameters		GD2000 drive electronical gear ratio			983Ta Z axis parameter		GD2000 drive electronical gear ratio	
			28	15	PA41	PA42		28	15	PA41	PA42
1	1	0.1	4	00000001	4	5		4	00010001	4	5
2	2	0.2	4	00010011	4	5		2	00010001	4	5
3	3	0.3	4	01000101	4	5		2	01000010	4	5
4	4	0.4	2	00010011	4	5		2	01100011	4	5
5	5	0.5	2	00010100	1	1		2	01100100	1	1
6	6	0.6	2	01000101	4	5		2	01010101	4	5
8	8	0.8	2	01100111	4	5		2	01110111	4	5
10	10	1.0	2	01101001	1	1		2	01111001	1	1

Para. NO.	Bit 7	Bit 6	Bit 5	Bit 4	Bit 3	Bit 2	Bit 1	Bit 0
0018					DIC		CPF2	CPF1

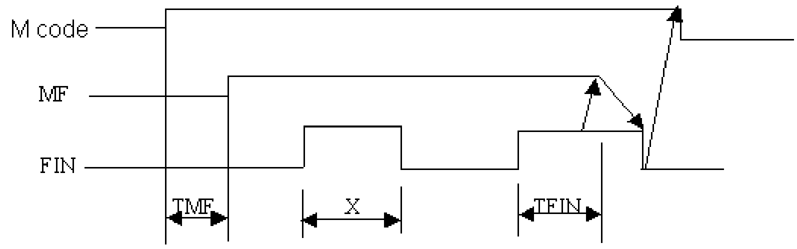
Detailed introduction

	Set as 1	Set as 0
DIC	The input unit is 1/10	The input unit is not 1/10
CPF2 CPF1	The backlash compensation pulse frequency (common to all axes) must be set as 256KHZ.	

Frequency KHZ	32	64	128	256
CPF2	0	0	1	1
CPF1	0	1	0	1

Para. NO.	Bit 7	Bit 6	Bit 5	Bit 4	Bit 3	Bit 2	Bit 1	Bit 0
0019	TMF				TFIN			

Detailed introduction		
	Set as 1	Set as 0
TMF	The time from sending M,S and T codes to the end of sending MF,SF,TF codes is 16~126ms (16ms as one interval).	
TFIN	The time of FIN signal receiving width is 16~256ms(16ms as one interval).	



Because X<TFIN, it is ignored.

TMF or TFIN=16×（N+1）ms (N=1-15)

TMF	TFIN	Setting			
16 m sec	More than 16 m sec	0	0	0	0
32 m sec	More than 32 m sec	0	0	0	1
48 m sec	More than 48 m sec	0	0	1	0
64 m sec	More than 64 m sec	0	0	1	1
80 m sec	More than 80 m sec	0	1	0	0
96 m sec	More than 96 m sec	0	1	0	1
112 m sec	More than 112 m sec	0	1	1	0
128 m sec	More than 128 m sec	0	1	1	1
144 m sec	More than 144 m sec	1	0	0	0
160 m sec	More than 160 m sec	1	0	0	1
176 m sec	More than 176 m sec	1	0	1	0
192 m sec	More than 192 m sec	1	0	1	1
208 m sec	More than 208 m sec	1	1	0	0
224 m sec	More than 224 m sec	1	1	0	1
240 m sec	More than 240 m sec	1	1	1	0
256 m sec	More than 256 m sec	1	1	1	1

Para. NO.	Bit 7	Bit 6	Bit 5	Bit 4	Bit 3	Bit 2	Bit 1	Bit 0
0020	CLSI						ZTNZ	ZTNX

Detailed introduction		
	Set as 1	Set as 0
CLSI	Not detect the servo position circuit LSI.	Detect the servo position circuit LSI
ZTNZ	X axis with the reference position return function	X axis without the reference position return function
ZTNX	Z axis with the reference position return function	Z axis without the reference position return function

<table>
<tr><td>Parameter No.</td><td>Bit 7</td><td>Bit 6</td><td>Bit 5</td><td>Bit 4</td><td>Bit 3</td><td>Bit 2</td><td>Bit 1</td><td>Bit 0</td></tr>
<tr><td>0024</td><td>PML2</td><td>PML1</td><td>DLME</td><td>RDAL</td><td></td><td></td><td></td><td></td></tr>
<tr><td colspan="9">Description</td></tr>
<tr><td></td><td colspan="4">Set to 1</td><td colspan="4">Set to 0</td></tr>
<tr><td>PML2
PML1</td><td colspan="8">Pitch error compensation override, the setting offset multiplied by this override is output.
<table><tr><td>override</td><td>×1</td><td>×2</td><td>×4</td><td>×8</td></tr><tr><td>PML2</td><td>0</td><td>0</td><td>1</td><td>1</td></tr><tr><td>PML1</td><td>0</td><td>1</td><td>0</td><td>1</td></tr></table>(for all axes)</td></tr>
<tr><td>DLME</td><td colspan="4">When one program is to be saved into memory, all the other programs saved in which before should be cleared auto.</td><td colspan="4">When one program is to be saved into memory, all the other programs saved in which before do not to be cleared auto.</td></tr>
<tr><td rowspan="2">RDAL</td><td colspan="8">Press O - 9 9 9 9 READ, the program is being storied.</td></tr>
<tr><td colspan="4">When storing a program, the memory saves them all.</td><td colspan="4">When storing a program, whether the memory saves just one or all of them is decided by MDI operation.</td></tr>
</table>

<table>
<tr><td>Parameter No.</td><td>Bit 7</td><td>Bit 6</td><td>Bit 5</td><td>Bit 4</td><td>Bit 3</td><td>Bit 2</td><td>Bit 1</td><td>Bit 0</td></tr>
<tr><td>0025</td><td>MUSR</td><td></td><td>MSUB</td><td>MPRM</td><td></td><td></td><td></td><td>TSE</td></tr>
<tr><td colspan="9">Description</td></tr>
<tr><td></td><td colspan="4">Set to 1</td><td colspan="4">Set to 0</td></tr>
<tr><td>MUSR</td><td colspan="4">User macro interruption function used.</td><td colspan="4">No user macro interruption function used.</td></tr>
<tr><td rowspan="2">MSUB</td><td colspan="4">Subprogram type of user macro interruption</td><td colspan="4">Macro program type of user macro interruption</td></tr>
<tr><td colspan="8">(Note) For subprogram type of user macro interruption, the local variable value is not changed before and after the interruption; while for macro program type of user macro interruption, the local variable is different after interruption.</td></tr>
<tr><td>MPRM</td><td colspan="4">M code controlling user macro program interruption is set by parameter (corresponding to parameter N053,N054)</td><td colspan="4">User macro program interruption is controlled by M96, and M97.</td></tr>
<tr><td rowspan="2">TSE</td><td colspan="4">State triggered interruption.</td><td colspan="4">Edge triggered interruption</td></tr>
<tr><td colspan="8">(Note)State triggered means interruption signal is valid in input state; edge triggered means interruption signal is valid when it is front edge.</td></tr>
</table>

Parameter No.	Bit 7	Bit 6	Bit 5	Bit 4	Bit 3	Bit 2	Bit 1	Bit 0

<table>
<tr><td>0026</td><td></td><td>FHDL</td><td>NGMP</td><td>OFFVY</td><td></td><td></td><td></td><td>CKIM</td></tr>
<tr><td colspan="9">Description</td></tr>
<tr><td></td><td colspan="4">Set to 1</td><td colspan="4">Set to 0</td></tr>
<tr><td>FHDL</td><td colspan="4">The MPG each scale amount is 0.01mm/0.001inch, influenced by no MP1, MP2 and NGMP signals.</td><td colspan="4">The MPG each scale amount is changed with MP1, MP2 signals (MP1~MP2: MPG override signal).</td></tr>
<tr><td>NGMP</td><td colspan="8">Move amount for MPG a scale</td></tr>
</table>

NGMP	MP2	MP1	Move amount metric(mm)	Move amount inch(inch)
0	0	0	0.001	0.0001
0	0	1	0.01	0.001
0	1	0	0.1	0.01
0	1	1	0.1	0.01
1	0	0	0.01	0.001
1	0	1	0.001	0.0001
1	1	0	0.1	0.01
1	1	1	0.1	0.01

	Set to 1	Set to 0
OFFVY	Have no servo alarm issued even VRDY is ON before PRDY output.	Servo alarm is issued when VRDY is ON before PRDY output.
CKIM	Auto run with no regard to machine lock signal.	Machine lock signal is active immediately.
	[Note] In MANUAL, machine lock is always active immediately.	

Parameter No.	
0027	CMRX
0028	CMRZ
Description	

Command overrides of X axis, Z axis respectively.

(1) When ACMR=0 (standard) in parameter No.0316, to set the value, please refer to Parameter No.014, and No.015.

Code setting	Override
1	0.5
2	1
4	2
10	5
20	10

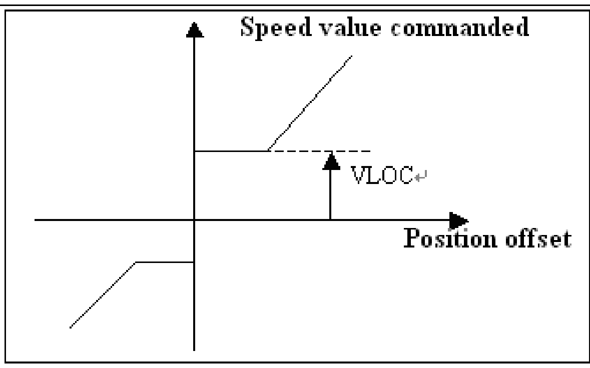
(Note) when the setting code is beyond the form and the override is 1, and then the parameter is set by decimal system. The standard code setting is 2 for pulse encoder.

(2)When ACMR=1 (any override) in parameter 0316.5,

$$CMR = \frac{\text{CMR num. para}}{\text{CMR denom. para}}$$

Setting range of command override CMR numerator parameter is 1~255;
Setting range of command override CMR denominator parameter is 1~255;
CMR numerator parameter: No.N027 of X axis and No.N028 of Z axis;
CMR denominator parameter: No. N333 of X axis and No.N334 of Z axis.

Parameter No.	
0031	VLOCX
0032	VLOCZ
Description	
VLOCX VLOCZ	The lowest speed clamping of X axis and Z axis respectively



setting range: 0~7(VELO)
speed value commanded(VELO)

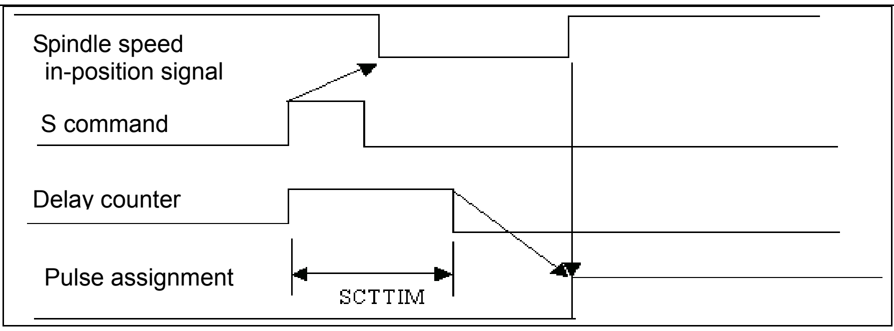
Parameter No.	
0035	MBUF1
0036	MBUF2
Description	
M code without buffering can be max. 2 (except M00,M01,M02 and M30). Setting range: 00~97	

Parameter No.	
0037	SPGST
Description	
SPGST	Spindle motor speed at spindle gear shift (for constant surface speed) Setting range: 0~255
Setting value = $\frac{\text{Motor speed at gear shift}}{\text{Spindle motor max. speed}} \times 4095$	

Parameter No.	
0038	SPSOR
Description	
SPSOR	Spindle speed at exact stop(for constant surface speed) Setting range: 0~255 unit: rpm

Parameter No.	
0040	EXOMAX
Description	
EXOMAX	In external input compensation B function, allowable value in limit check (disabled).
Setting range: 0~255 Unit: 0.002 mm (metric input) / 0.0002 inch (inch input)	

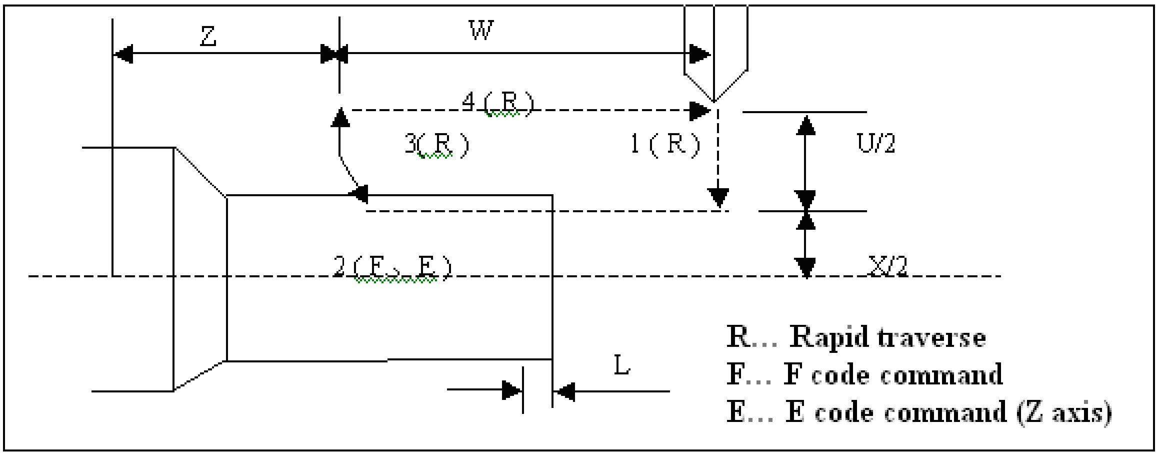
Parameter No.	
0062	SCTTIM
Description	
SCTTIN	Delay time setting of spindle speed arriving signal detection, and time setting from executing S function to spindle in-position signal detection. Setting range: 0~255, unit: msec



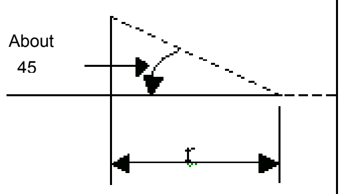
Parameter No.	
0041	EXOFS1
0042	EXOFS2
0043	EXOFS3
0044	EXOFS4
0045	EXOFS5
0046	EXOFS6
0047	EXOFS7
0048	EXOFS8
0049	EXOFS9
0050	EXOFS10
0051	EXOFS11
0052	EXOFS12
0053	EXOFS13/MACINTON
0054	EXOFS14 MACINTOF
0055	EXOFS15
0056	EXOFS16
Description	
EXOFS1 ~ EXOFS16	In external input compensation B function, override for outer diameter check (disabled) setting range: 0~15 unit: 0.1
MACINTON	M code making user macro program interruption valid setting range: 03~97
MACINTOF	M code making user macro program interruption invalid setting range: 03~97
[Note] The MACINTON, MACINTOF of parameter No.053, 054 are only active when the 4th bit of parameter 025 is 1. Otherwise, they are inactive even set.	

Parameter No.	
0057	TMHOR(h)
0058	TMMIN(min)
0059	TMSEC(s)
Description	
TMHOR(h)	Time display 0~255 (1h increment)
TMMIN(min)	Minute 0~59 (1min increment)
TMSEC(s)	Second 0~58 (2s increment)
Process time (time for STL lighting) displayed by hour, minute, and second, is saved in RAM even power off, but because RAM is written into every 6 min, data less than 6 min will be cleared with the power shut. To preset to 0, set as normal.(also can make it by pressing [set] key to change its value)	

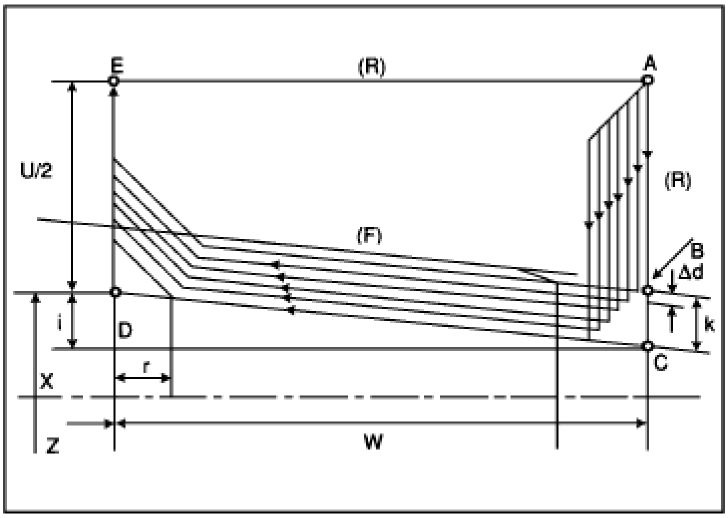
Parameter No.	
0064	THDCH
Description	
THDCH	Chamfering width in thread cutting (G92, G76). Setting range: 0~127 (unit: 0.1 pitch), which can be set by MDI panel.



Due to the delay of servo system, the chamfering showing as the left is less than or equal to 45º.



Threading chamfering
Threading cycle (G76)



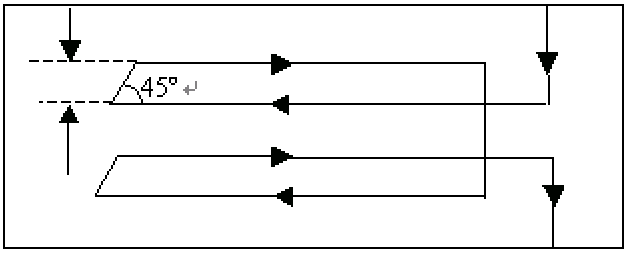
r: chamfering part in thread cutting

Parameter No.	
0065	GROVE
Description	
GROVE	Return of G74 and G75
setting range: metric input 0~16383 unit: 0.001mm(by radius) inch input 0~16383 unit: 0.0001inch (also can make it by pressing [set] key to change its value)	

Parameter No.	
0066	THDFN
Description	
THDFN	Finishing allowance of G76

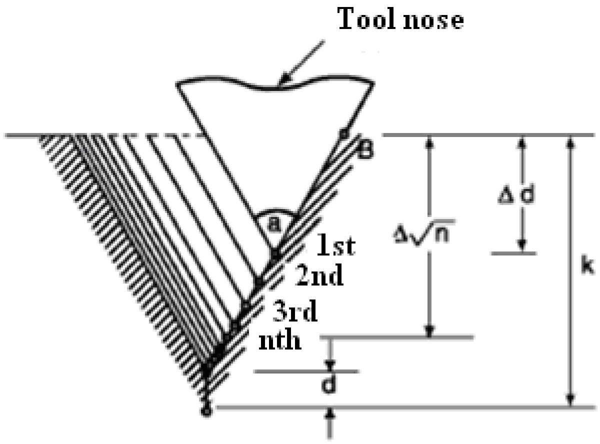
setting range: metric input 0~16383 unit:0.001mm (by radius) inch input 0~16383 unit: 0.0001inch All the value above is specified by radius. This parameter can be set by MDI panel (ADDRESS SET)---refer to No.68 parameter. (also can make it by press[set] key to change its value)

Parameter No.	
0067	MRCDT
Description	
MRCDT	Retraction amount of compound canned cycle G71 and G72.
setting range: metric input 0~16383 unit: 0.001mm (specified by radius) inch input 0~16383 unit: 0.0001inch (specified by radius) (also can make it by press[set] key to change its value) cutting feed:	



Parameter No.	
0068	THCLM
Description	
THCLM	Min. cut depth of thread cutting cycle G 76

setting range: metric input0~16383 unit: 0.001mm
inch input0~16383 unit: 0.0001inch
(also can make it by press[set] key to change its value)

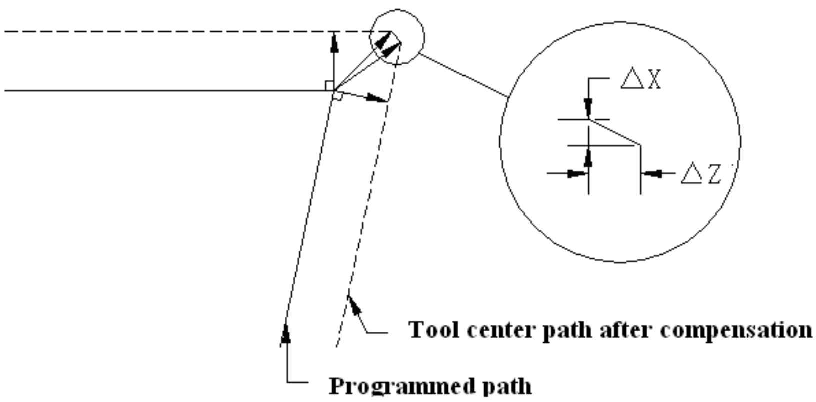


If the result calculated by formula $d(\sqrt{n+1}-\sqrt{n})$ is less than THCLM value, then the clamping position of the cutting is the THCLM value.

Parameter No.	
0069	CRCDL

Description	
CRCDL	The minor move limit for neglecting at outer acute angle near 90º in tool nose radius compensation

The minor move is ignored when △X<CRCDL, △Z<CRCDL.
By this process, it may eliminate the affection to the workpiece by the tool.



Parameter No.	
0070	INPX

Appendix 5 Parameters

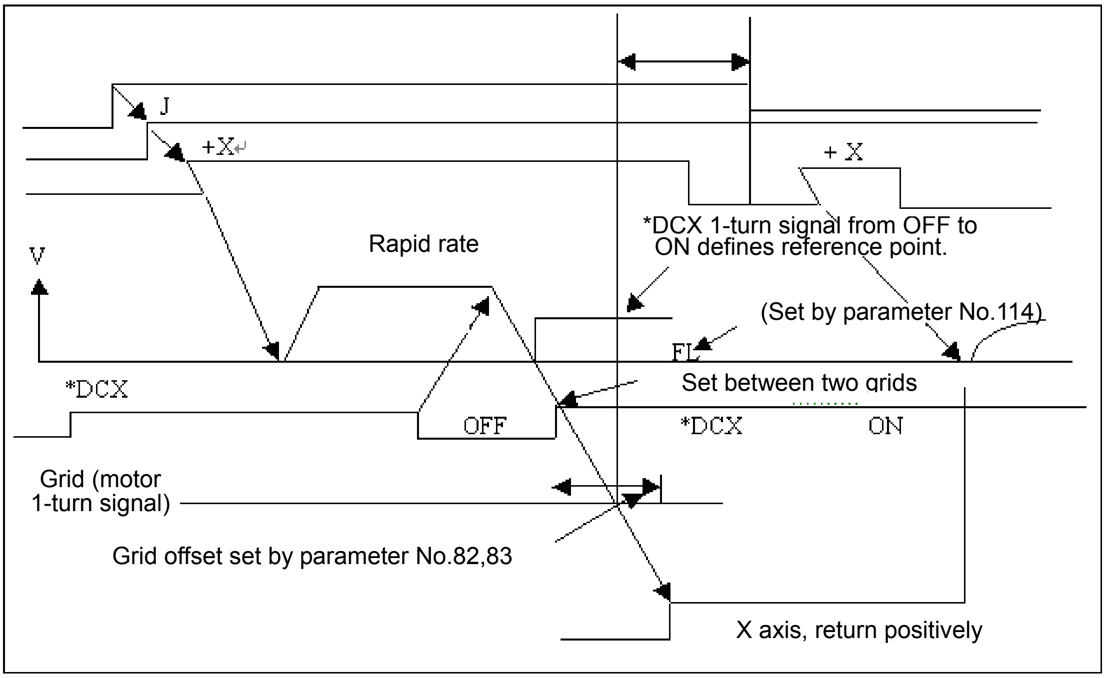
0071	INPZ
Description	
INPX INPZ	In-position width settings of X axis and Z axis respectively Setting range: 0~32767 detection unit, standard setting: 10.

Parameter No.	
0074	STPEX
0075	STPEZ
Description	
STPEX STPEZ	Position offset limit at stopping of X axis and Z axis respectively

Parameter No.	
0078	SERRX
0079	SERRZ
Description	
SERRX SERRZ	Position offset limit at moving of X axis and Z axis respectively setting range: 0~32767 detection unit

Parameter No.	
0082	GRDSX
0083	GRDSZ
Description	
GRDSX GRDSZ	Grid offset of X axis and Z axis respectively (only for grid). See to parameter No.14 and No.15) Setting range: 0~±32767 detection unit. Reference point moving positively (negatively), the value is set positive (negative), which should not beyond the capacity of reference counter.

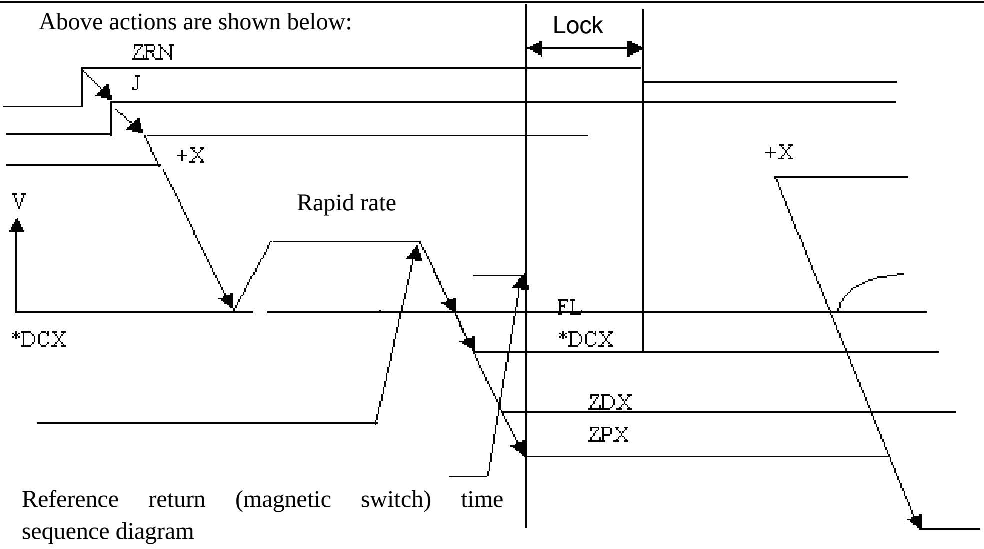
(1)reference point return in grid mode



Select manual consecutive feed mode (JOG), connect ZRN signal (connected with +0N), feed to the reference point by JOG button and the mobile part of the machine tool will move rapidly, slow down but keep moving in a low speed when meeting the deceleration limit switch (which give deceleration signals *DCX and *DCZ in the return to reference point) and finally stop and send return end signals ZPX, ZPZ when arriving the fist grid position after deceleration limit switch reset. About the directions after reference point return, they can be set by all axes that once reference return completed and ZPX, ZPZ signals sent, no manual feed instructions can be executed before ZRN signal cut.

(2)reference point return in magnetic switch mode

Above actions are shown below:



Reference return (magnetic switch) time sequence diagram

So called reference point return in magnetic switch mode, means that in the action (1), after rising edges of ZDX and ZDZ signals which replace the grid signal emerge, the feed stops and reference point rerun end signals ZPX and ZPZ are issued. Moreover, the reference point return detection cannot be performed by G27.

Parameter No.	
0086	LPGMX
0087	LPGMZ
Description	
LPGMX LPGMZ	Servo loop gain constant setting of X, Z axes

$$\text{Setting value} = 2048 \times \frac{E}{L}$$

E= 7V(7V/1000rpm motor); 3.5 V(7V,2000r/min motor)

L = Mechanical move amount for motor one-turn mm or inch

a = detection unit(mm or inch)

eg: L = 2mm E=7V(motor 1000r/min,7v)

setting value: 2048x7/2x0.0005x1000=3584 (detection unit 0.0005mm)

Machine tool feed amount for motor per turn	axis	Loop gain series	
		7V,1000r/min servo motor	7V,2000r/min servo motor
10mm,10deg	X & Z	1434	717
8mm,8deg	X & Z	1792	896
6mm,6deg	X & Z	2389	1195
5mm,5deg	X	2867/1434	1434/717
	Z	2867	1434
4mm,4deg	X	3584/1792	1792/896
	Z	3584	1792
3mm,3deg	X	4779/2389	2389/1195
	Z	4779	2389
2mm,2deg	X	7168/3584	3584/1792
	Z	7168	3584
1mm,1deg	X	7168/3584	3584/1792
	Z	3584	3584
0.5inch	X	2867/1433	1434/717
	Z	2867	1434
0.4 inch	X	3584/1792	1792896
	Z	3584	1792
0.3 inch	X	4779/2389	2389/1195
	Z	4779	2389
0.25 inch	X	5734/2867	2867/1437
	Z	2867	2867
0.2 inch	X	7168/3584	3584/1792
	Z	7168	3584
0.15 inch	X & Z	4778	2389
0.1 inch	X & Z	7168	3584

To X axis, the left value of / sign is for radius programming, right one for diameter programming. Sheet above is standard and the max. Feedrate limit value should be considered if DMR and CMR unchanged.

Parameter No.	

	0	9	1

0090	LPGIN
Description	
LPGIN	Position control loop gain, setting range: 1~9999, unit: 0.01 sec^{-1}

JOGF

JOGF JOG feedrate of the override switch lying 100% spot

setting range: 1～150 unit: mm/min, deg/min (mm output)

1～60 unit: 0.1inch/min, 0.1deg/min (inch output) or 1deg/min (inch output)

standard setting: 20

0091	JOGF

JOGF JOG feedrate of the override switch lying 10 spot

Setting range: 1~150 unit: mm/min (metric output);

1~ 60 unit: 0.1inch/min (inch output)

Eg: Set 20mm/min to Parameter No.91

spot	Feedrate override	Manual consecutive feed	
		MM	INCH
0	0%	0mm/min	0 inch/min
1	10	1.0	0.02
2	20	1.4	0.03
3	30	2.0	0.04
4	40	2.7	0.06
5	50	3.7	0.08
6	60	5.2	0.10
7	70	7.2	0.14
8	80	10	0.2
9	90	14	0.3
10	100	**20**	**0.4**
11	110	27	0.6
12	120	37	0.8
13	130	52	1.0
14	140	72	1.4
15	150	100	2
16	160	140	3
17	170	200	4
18	180	270	6
19	190	370	8
20	200	520	10

21	200	720	14
22	200	1000	20
23	200	1400	30
24	200	2000	40

Note1: the federate take 100% feedrate as the base and change in geometric series. And other feedrates can be got by setting 100% federate through parameter.

Note2: in sheet above, the speed error is ±3%.

Parameter No.	
0092	RPDFX
0093	RPDFZ
Description	
RPDFX RPDFZ	Rapid traverse rate of X axis and Z axis respectively Setting range: 30~24000 unit: mm/min (metric output) 30~9600 unit: 0.1inch/min (inch output)

Parameter No.	
0096	LINTX
0097	LINTZ
Description	
LINTX LINTZ	Time constant of linear acceleration and deceleration(for rapid)of X axis and Z axis respectively setting range: 8~4000 unit: ms

Parameter No.	
0100	EXPTX
0101	EXPTZ
Description	
EXPTX EXPTZ	Index acceleration and deceleration time constant of X axis and Z axis JOG feed respectively setting range: 8~4000 unit: ms

Parameter No.	
0104	THRDT
Description	
THRDT	Use index acceleration and deceleration time constant of X axis in thread cutting with No. 107 parameter to set most proper value. setting range:1~4000 unit: msec

(Note) when time constant of No.104 parameter is small, the retraction of the thread can be small and the retraction time can be shortening in thread chamfering. However, too small value will lead to mechanical shock. So by setting time constant of parameter No.104 and acceleration and deceleration lower limit feedrate of No. 107 parameter, while thread chamfering is performing, the retraction of the thread can still be small though this parameter value is larger. Usually No.104 parameter(time constant) should refer to the max feedrate of thread cutting and No.107 parameter(lower limit feedrate)should be 2m/min.

Parameter No.	
0105	FEEDT
Description	
FEEDT	Index acceleration and deceleration time constant in cutting feedrate setting range: 8~4000 unit: ms

Parameter No.	
0106	FEDMX
Description	
FEDMX	Upper limit of cutting feedrate(for all axes) setting range: 6~15000 unit: mm/min (metric output) 6~6000 unit: 0.1inch/min (inch output)

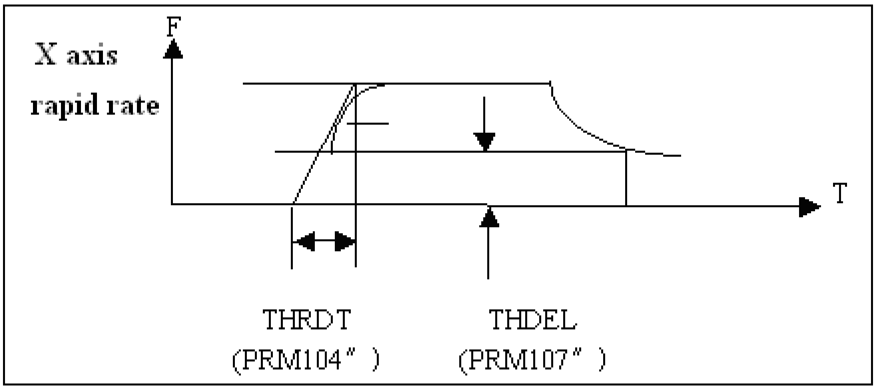
Parameter No.	
0107	THDFL
Description	
THDFL	Lower limit of X axis index acceleration and deceleration(FL)in thread cutting

setting range: 6~15000　unit: mm/min (metric output)
　　　　　　　6~6000　 unit: 0.1inch/min (inch output)

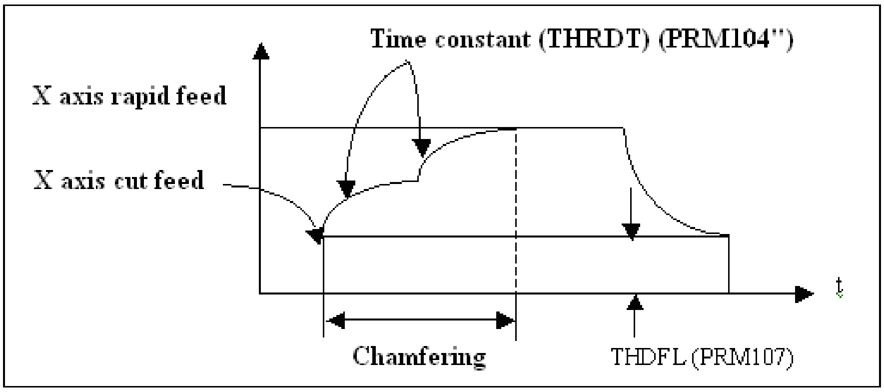
When this parameter is large enough, the retraction of the thread can be small and in thread chamfering, tool retraction time can be shorten. However if the value is too large, mechanical shock may occurs. So please choose proper value according to the machine situation.

standard setting: 2000mm/min　　　　　　　　　　(refer to parameter No.104)

（1）　In thread chamfering



(2)perform thread chamfering



confirm after parameter setting

After the parameter setting, in cutting thread operation(G92) mode, dry run at max feedrate in thread cutting and confirm the following items in thread chamfering/ non-chamfering situations respectively:

(1) Detect the wave of X axis TSA in thread chamfering and make sure that overshooting is within 5%.

(2) Ensure that the shock to machine is not huge.

Parameter No.	
0108	FEDFL
Description	
FEDFL	Lower limit of　index acceleration and deceleration in cutting feedrate(FL) setting range: 6~15000　unit: mm/min (metric output) 6~6000　 unit 0.1inch/min (inch output) This value is always set to 0.

Parameter No.	
0109	JGFLX
0110	JGFLZ
Description	
JGFLZ	Lower limit of index acceleration and deceleration of X axis and Z axis in manual consecutive feedrate(FL) setting range: 6~15000 unit: mm/min(metric output) 6~6000 unit: 0.1inch/min(inch output)

Parameter No.	
0113	RPDFL
Description	
RPDFL	Min. feedrate of rapid override (F0) (all axes share setting range: 6~15000 unit: mm/min (metric output) 6~6000 unit: 0.1inch/min(inch output)

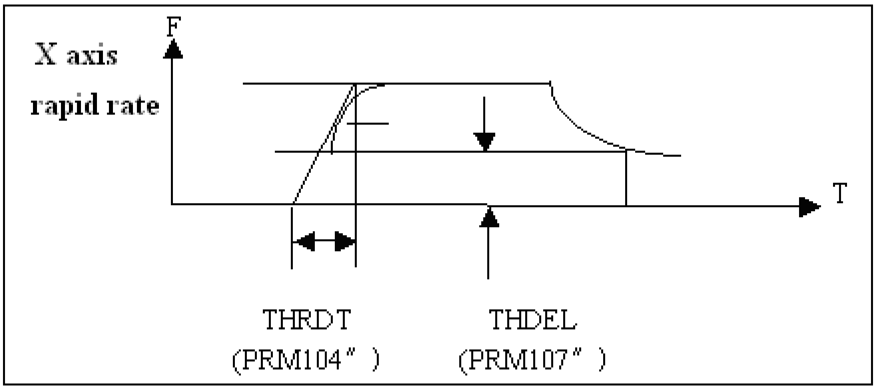
Parameter No.	
0107	THDFL
Description	
THDFL	Lower limit feedrate of X axis index acceleration and deceleration in thread cutting(FL)

setting range: 6~15000 unit: mm/min(metric output)
6~6000 unit0.1inch/min(inch output)

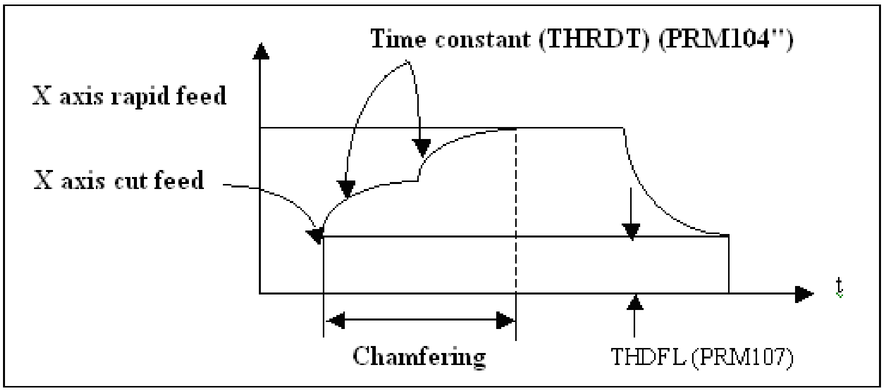
When this parameter is enough large, the retraction of the thread can be small and in thread chamfering, tool retraction time can be shorten. However if the value is too large, mechanical shock may occurs. So please choose proper value according to the machine situation.

standard setting:2000m/min (refer to parameter No.104)

（1） In thread chamfering



(2)perform thread chamfering



confirm after parameter setting

After the parameter set, in cutting thread operation(G92) mode, dry run at max feedrate in thread cutting and confirm the following items in thread chamfering/ non-chamfering situations respectively:

(1) Detect the wave of X axis TSA in thread chamfering and make sure that overshooting is in 5%.

(2) Ensure that the shock to machine is not huge.

Parameter No.	
0114	ZRNFL
Description	
ZRNFL	Low federate in reference point return(FL) (all axes share) setting range: 6~15000 unit: mm/min (metric output) 6~6000 unit: 0.1inch/min (inch output)

Parameter No.	
0115	BKLX
0116	BKLZ
Description	
BKLX BKLZ	Backlash of X and Z axis respectively(when it is diameter specifying, X axis is set by diameter)
setting range: 0~225 unit: 0.001mm (metric output) 0~225 unit: 0.0001 inch (inch output)	
(Note) when ACMR=1(any instruction detection override) in PRM316, backlash unit is detection unit.	

Parameter No.	
0119	SPDLC
Description	
SPDLC	Spindle speed deviation compensation, that is, set the compensation of speed instruction voltage zero offset.(used in constant control B) setting range: 0~±8191 unit: VELO

Parameter No.	
0124	DRFTX
0125	DRFTZ
Description	
DRFTX DRFTZ	Drift compensation in servo loop of X and Z axis respectively
setting range: 0~±500 unit: VELO This value will auto change after auto drift compensation parameter set(ADFT of No.006) [Note]When the compensation is over 500, No. 412 or No.422 alarm will occur.	

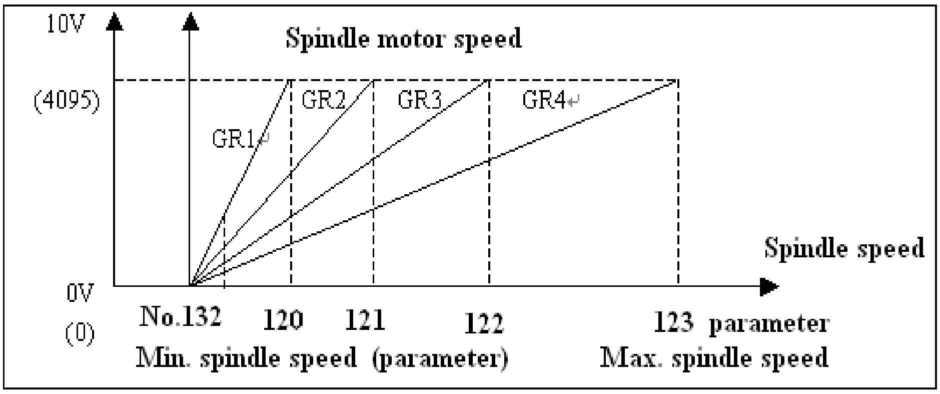
Parameter No.	
0132	LOWSP
Description	
LOWSP	Spindle min. speed (G96) in constant surface speed control only valid when the fifth bit of No.307 parameter SLOW=1 setting range: 0~9999 unit: rpm

Parameter No.	
0133	ACALFL
Description	

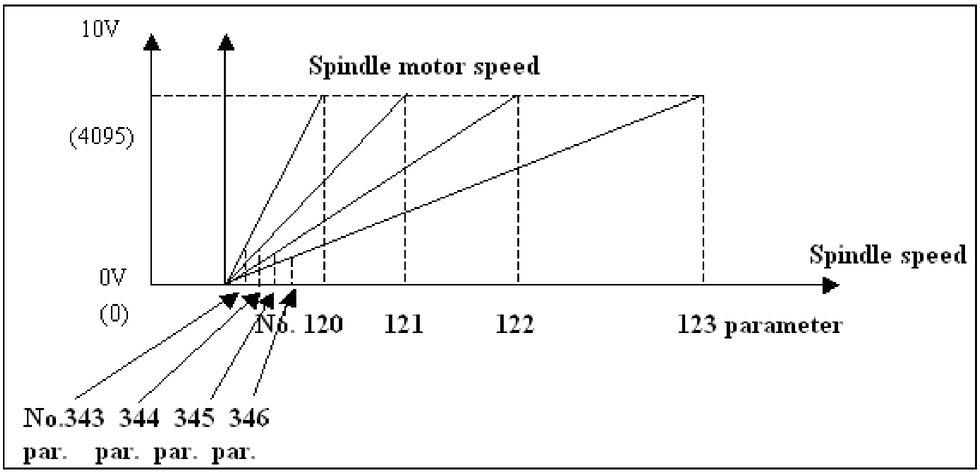
ACALFL	Measure speed with auto tool compensation function(apply to all axes) setting range: 6~15000mm/min 6~6000inch/min

Parameter No.	
0120	GRMX1
0121	GRMX2
0122	GRMX3
0123	GRMX4
Description	
GRMX1~GRMX4 The max. number of spindle revolutions corresponding to gear 1~4(S code is 4095 in binary output, revolution number when 10V in S4 analog output)(only valid with peripheral speed constant arbitrary control function) setting range: 1~32767 (unit: RPM)	

(1) For gear 1~4,the min. spindle revolution number are the same(Parameter No.307 fifth bit SLOW=1)



(2) when to gear 1~4,the min. spindle revolution number are not the same (Parameter No.307 fifth bit SLOW=0)



Parameter No.	
0134	SPDMX
Description	
SPDMX	Spindle max speed limit setting range: 0~9999 unit r/min
[Note] if it is set to 0,then no clamping to spindle speed	

Parameter No.	
0140	PSANGN
Description	
PSANGN	Data for gain adjustment of S analog output
Range 700~1250 Standard setting value1000 (adjust method) （1） Set the standard setting value“1000” （2） Specify the max value of S analog quantity (10V) （3） Detect output voltage （4） Set PSANGN again according to the following formula $\frac{10.0}{\text{Measure voltage(V)}} \times 1000$=set value （5） After setting parameter, then specify S analog quantity is the max.(10V), finally ensure the output voltage is 10V	

Parameter No.	
0141	TIMEI
Description	
TIMEI	Preset the using time and operation can preset too. setting range: 0~32767 unit: 0.1h

Parameter No.	
0142	TIME2
Description	
TIME2	Preset the using time setting range: 0~99999999 unit: 0.1h

Parameter No.	
0143	LT1X1
0144	LT1Z1

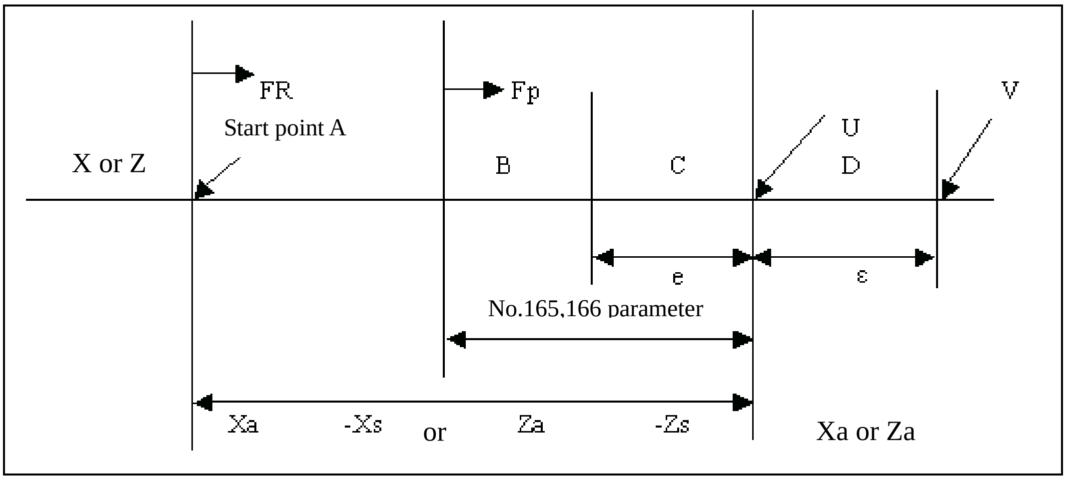
0147	LT1X2
0148	LT1Z2
0151	LT2X1
0152	LT2Z1
0153	LT2X2
0154	LT2Z2
0155	LT3X1
0156	LT3Z1
0157	LT3X2
0158	LT3Z2
Description	
LT1X1 LT1Z1 LT1X2 LT1Z2 LT2X1 LT2Z1 LT2X2 LT2Z2 LT3X1 LT3Z1 LT3X2 LT3Z2	LT□□□ 1 No. n vertex in four corner area Coordinate axis No. n stroke limit 2 Set upper limit of the stroke setting range: 0~±99999999 unit: 0.001mm (metric output) 0~±99999999 unit: 0.0001 inch (inch output) Among then no.151~158 also can be used in setting operations (when it is diameter specifying, X axis is specified by diameter).

Parameter No.	
0159	REF2X
0160	REF2Z
Description	
REF2X REF2Z	Distance from 2nd reference point o to 1st reference point X or Z axis respectively. When it is diameter specifying, X axis is specified by diameter. setting range: 0~±99999999 unit: 0.001mm (metric output) 0~±99999999 unit: 0.0001inch (inch output)

Parameter No.	
0163	GANMAX
0164	GANMAZ
Description	
GANMAX	r values of X and Z axes with auto tool offset function

GANMAZ	setting range: 1~±99999999 unit: 0.001mm(metric output) 1~±99999999 unit: 0.0001inch(inch output)
[Note1] X axis is specified by radius. [Note2] set data should larger than no.165, 166 parameter values.	

Parameter No.	
0165	EPCX
0166	EPCZ
Description	
EPCX EPCZ	ε values of X and Z axes with auto tool offset function. X axis is specified by radius. setting range:1~±99999999 unit: 0.001mm 1~±99999999 unit: 0.0001inch Note: X axis is specified by radius.



FR: rapid feedrate

FP: measuring feedrate(see to Parameter No.133)

In G36 or G37, according to move command (Xa or Za), machine tool moves at rapid federate in the area A from starting point to the preset measuring point. After it stops at T point, it will move at the measuring feedrate as No.133 parameter to area B, C, D, during when, if it issues measuring point arrival signal, it will stop at once. If measuring point arrival signal is issued beyond area C and D or still not issued when machine tool arrives V point, no.80 alarm occurs.

Parameter No.	Bit 7	Bit 6	Bit 5	Bit 4	Bit 3	Bit 2	Bit 1	Bit 0
0306	SKPF				NEOP		TMCR	OFM

Description

	Set to 1	Set to 0
SKPF	In skip function command (G31),feedrate is FL, set by parameter no.342.	In skip function command (G31), feedrate is specified by F code.

NEOP	When program is being saved in memory, M02, M03, M99 are not program closure.	program closure
TMCR	T code used in user macro program	T code can't be called
OFM	Tool position compensation valid in blocks with move commands	Tool position compensation always valid

Parameter No.	Bit 7	Bit 6	Bit 5	Bit 4	Bit 3	Bit 2	Bit 1	Bit 0
0307	FCUT	ABIC	SLOW					OTCS

Description		
	Set to 1	Set to 0
FCUT	In compound fixed cycle (G71,G72), finish machining doesn't be performed in rough machining	In compound fixed cycle (G71,G72), finish machining be performed in rough machining
ABIC	[ABS/INC] key is valid when X or Z key input in MDI and special G code B or C used.	[ABS/INC] key is invalid when X or Z key input in MDI and special G code B or C used.
SLOW	S 4-bit code and 12-bit binary code output or low speed clamping position of analog output is valid to all gears.	S 4-bit code and 12-bit binary code output or low speed clamping position of analog output is valid to gears respectively.
OTCS	Machine tool immediately stops when mobile parts have touched machine limit switch. (machine position is lost)	Machine tool decelerates and stops when mobile parts have touched machine limit switch. (machine position isn't lost)

Parameter No.	Bit 7	Bit 6	Bit 5	Bit 4	Bit 3	Bit 2	Bit 1	Bit 0
0308	DIOM	MSFT		MANP	RSTB			

Description		
	Set to 1	Set to 0
DIOM	Macro program variable can write and read DI and DO	Can't write and read DI and DO
MSFT	When there is user macro program,[SHIFT] key is valid from keyboard	[SHIFT] key invalid from keyboard
MANP	When user macro program independent variable with no decimal point, it will be dealt as integer.	When user macro program independent variable with no decimal point, however as the rule, it will be dealt as the value with the decimal point

RSTB	Use emergency stop, outside reset, reset and rewinding to reset with no resetting signal output	Output reset signal

Parameter No.	Bit 7	Bit 6	Bit 5	Bit 4	Bit 3	Bit 2	Bit 1	Bit 0
0309	TLSK	GRST		LCTM			APZ	APX

Description		
	Set to 1	Set to 0
TLSK	Input group no. in tool skip	Input no group no.
GRST	Clear all execution data in storage groups for reset signal input	Clear all execution data in groups in which tool life expired for reset signal input
LCTM	Tool life specified by time	Tool life specified by times
APZ APX	Set auto coord. sys. of X and Z axis(select)valid/invalid respectively	
	Auto coord. sys. setting valid	invalid

Parameter No.	Bit 7	Bit 6	Bit 5	Bit 4	Bit 3	Bit 2	Bit 1	Bit 0
0310	NFED1		RSCB1	STP21	BAD1			
0311	NFED2		RSCB2	STP22	BAD2			
0312	NFED3		RSCB3	STP23	BAD3			
0313	NFED4		RSCB4	STP24	BAD4			

Description		
	Set to 1	Set to 0
NFED 1,2,3,4	When input/output device is used, they decide whether initial and terminal pilot holes and blank spaces between programs are output or not respectively.	
	No pilot holes and blank spaces output	output
RSCB 1,2,3,4	When input/output device 1,2,3,4 is used, control code(DO1~DO4) is used or not respectively	
	No control code used	Control code used
STP2 1,2,3,4	When input/output device 1,2,3,4 is used, set 2-bit or 1-bit stop respectively	
	2-bit stop	1-bit stop
BAD 1,2,3,4	Set the baud rate of input/output device 1,2,3,4 respectively	

Baud rate	BAD1,2,3,4			
50	0	0	0	0
100	0	0	0	1
110	0	0	1	0
150	0	0	1	1
200	0	1	0	0
300	0	1	0	1
600	0	1	1	0
1200	0	1	1	1
2400	1	0	0	0
4800	1	0	0	1
9600	1	0	1	0

[Note] with ROBOT interface select, baud rate of data transmitting between NC and ROBOT is set BAD4

[Note]see to parameters 340 and 341

Parameter No.	Bit 7	Bit 6	Bit 5	Bit 4	Bit 3	Bit 2	Bit 1	Bit 0
0314		MINT					HZ	HX

Description		
	Set to 1	Set to 0
MINT	Begin to execute interruption program after current block executed (user macro program interruption II)	Execute interruption program immediately(user macro program interruption)
HZ HX	Set hand wheel interruptions of X axis and Z axis respectively	
	Valid	Invalid

Parameter No.	Bit 7	Bit 6	Bit 5	Bit 4	Bit 3	Bit 2	Bit 1	Bit 0
0315	PRT	FCSS						

Description		
	Set to 1	Set to 0
PRT	Output nothing to leading zero when DPRNT instruction is used to output data	Output blank space to leading zero when DPRNT instruction is used to output data (DPRNT: outside data output instruction)
FCSS	S analog voltage output is changed by 8ms time interval.(new specification)	S analog voltage output is changed by 64ms time interval.(old specification)

Parameter No.	Bit 7	Bit 6	Bit 5	Bit 4	Bit 3	Bit 2	Bit 1	Bit 0
0316	CDSCG	PCFBK	ACMR				DSCGZ	DSCGX

Description		
	Set to 1	Set to 0

CDSCG	No frequency detection of DSCG feedback(resolver and inductosyn)	Do detection (this parameter is always set to 0 after initial adjustment)
PCFBK	Do servo feedback detection	No servo feedback detection(see to parameters 363 and 364)
ACMR	Set special CMR is permitted(not recommended)	Not allowed to set special CMR

Parameter No.	Bit 7	Bit 6	Bit 5	Bit 4	Bit 3	Bit 2	Bit 1	Bit 0
0318	PRG9	MSC9	MPD9				NSRH	RSTL

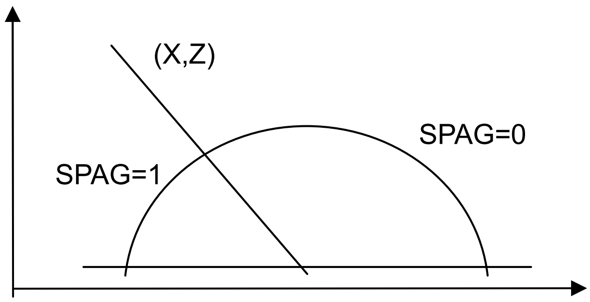
Description

	Set to 1	Set to 0
PRG9	Can't edit programs prg no.9000-9899	Can edit programs prg no.9000-9899
MSC9	When any prg no.9000-9899 program is being executed, and in single block mode, user macro program statement executing stops in single program block too.	When any prg no.9000-9899 program is being executed, even in single block mode, user macro program statement executing does not stop.
MPD9	Do not show the contents when a prg no.9000-9899 program is being executed.	Show the contents when a prg no.9000-9899 program is being executed.
NSHR	Don't output OP signal in sequence no. retrieval.	Output OP signal in sequence no. retrieval.
RSTL	Don't output STL signal when a program is saved in the memory by cycle start button in edit mode.	Output STL signal when a program is saved in the memory by cycle start button in edit mode.

Parameter No.	Bit 7	Bit 6	Bit 5	Bit 4	Bit 3	Bit 2	Bit 1	Bit 0
0319	PRG8	MSC8	MPD8				SPAG	MCS7

Description

	Set to 1	Set to 0
PRG8	Can't edit prg no.8000-8999 program	Can edit prg no.8000-8999 program
MSC8	When any prg no.8000-8999 program is being executed, and in single block mode, user macro program statement executing stops in single program block too.	When any prg no.8000-8999 program is being executed, even in single block mode, user macro program statement executing does not stop.
MPD8	Do not show the contents when a prg no.8000-8999 program is being executed.	Show the contents when a prg no.8000-8999 program is being executed.

SPAG	When directly input (session programming) in figure size, supplementary angle is used in angle instruction.	When directly input (session programming) in figure size, regular specification is used in angle instruction.
		
MCS7	When any prg no. 0001-7999 program is being executed, and in single block mode, user macro program statement executing stops in single program block too.	When any prg no. 0001-7999 program is being executed, even in single block mode, user macro program statement executing does not stop.

Parameter No.	
0320	UMMCD1(corresponding to prg no.9001)
0321	UMMCD2(corresponding to prg no.9002)
0322	UMMCD3(corresponding to prg no.9003)

UMMCD1, 2, 3; M code calling user macro program (at most set 3 of them).
Setting range: 01-97
(Can't call user macro program by M00,even 00 is set)

Parameter No.	
0323	UMGCD0(corresponding to prg no.9010)
0324	UMGCD1(corresponding to prg no.9011)
0325	UMGCD2(corresponding to prg no.9012)
0326	UMGCD3(corresponding to prg no.9013)
0327	UMGCD4(corresponding to prg no.9014)
0328	UMGCD5(corresponding to prg no.9015)
0329	UMGCD6(corresponding to prg no.9016)
0330	UMGCD7(corresponding to prg no.9017)
0331	UMGCD8(corresponding to prg no.9018)
0332	UMGCD9(corresponding to prg no.9019)

UMGCD0, 1, 2, 3,…, 9; G code calling user macro program (at most set 10 of them).
setting range: 001-255
(Can't call user macro program by G00,even 00 is set)

Parameter No.	
0336	PECZER

0337	PECZRZ

PECZRX, Z: pitch error origins of X axis and Z axis. setting range: 0-127
According to this parameter set given point corresponding to reference point.
(Eg) pitch error origin is set to 0, given point 1 is in +8.000mm, and given point 127 is in +1016.000mm, so the compensation range is 0-+1016.000mm.

0(pitch error origin)

1	2	3	127

0(machine origin) +8.000mm +16.000mm ...... +1016.000

1st compensation value is set to system parameter No.1001 (X axis) or No.2001 (Z axis)
2nd compensation value is set to system parameter No.1002 (X axis) or No.2002 (Z axis)
3rd compensation value is set to system parameter No.1003 (X axis) or No.2003 (Z axis)
....

This example shows situation when pitch error compensation interval (parameter379, 380) is 8000.

Parameter No.	
0340	IDVICE
Description	
IDVICE	Program being saved into the memory, set the input device selection(when in “set” interface INPUT DEVICE2=1,it means the setting is valid): standard setting value: 2
setting value is 2: RS232C(baudrate etc. should be set in parameter No.311) setting value is 3: RS232C(baudrate etc. should be set in parameter No.312)	

Parameter No.	Bit 7	Bit 6	Bit 5	Bit 4	Bit 3	Bit 2	Bit 1	Bit 0
0341	ODVICE							
Description								
ODVICE	Output device selection. Standard setting value:2							
setting value is 2: RS232C(baudrate etc. should be set in parameter No.311) setting value is 3: RS232C(baudrate etc. should be set in parameter No.312)								

Parameter No.	
0342	PSKPFL
Description	

PSKPFL	FL speed of skip cutting(applied to all axes) setting range: 6~15000 unit: 1mm/min (inch output)

Parameter No.	
0343	GRMIN1
0344	GRMIN2
0345	GRMIN3
0346	GRMIN4
Description	
GRMIN 1~4	4 gear shift, minimum spindle rotation number in constant surface control (G96) mode Setting range: 0~9999 unit: RPM Only valid when parameter No.307 SLOW=0.

Parameter No.	
0363	PCFBKX
0364	PCFBKZ
Description	
PCFBKZ PCFBKZ	Move amounts of X axis and Z axis in servo feedback detection
[Note1] Servo feedback detection is not performed as it is set to 0. [Note2] The move amount of servo feedback detection is fixed at 0.255mm (0.025inch). [Note3] the setting value is machine movement, so it should be small as possible but not so small to alarm. Make sure to set a proper one.	

Parameter No.	
0375	PPRTMX
0376	PPRTM Z
Description	
PPRTMX PPRTMZ	Values of X axis and Z axis respectively, set by auto coordinate system in metric input mode. The distance from the set origin point of the coordinate system to the 1st reference point is set by metric. setting range: 0~99999999 unit: 0.001mm With inch/metric switch select, parameter 379, 380 should be set too. Only valid for those axes set in auto coordinate system by parameter 309.

Parameter No.	
0379	PPRT1X

Appendix 5 Parameters

0380	PPRTIZ
Description	
PPRT1X PPRTIZ	Values of X axis and Z axis respectively, set by auto coordinate system in inch input mode. The distance from the set origin point of the coordinate system to the 1st reference point is set by inch. Setting range: 0~99999999 unit0.0001inch. With inch/metric switch select, parameter 375, 376 should be set too. Only valid for those axes set in auto coordinate system by parameter 309.

Parameter No.	
0383	PECINTX
0384	PECINTZ
Description	
PECINTX PECINTZ	Intervals setting of X axis and Z axis pitch error compensation respectively. setting range: 8000~20000000 unit: 0.001mm (metric input) 4000~20000000 unit: 0.0001 inch (inch input) (Max compensation interval=set interval×127)

Parameter No.	
0387	Program password
Description	
Program password	A secret digits stored in advance for program lock (not displayed, only for the program after the No.9000) Setting range:1~99999999

Parameter No.	
0408	LOCK / UNLOCK
Description	
LOCK / UNLOCK	When value input is identical with that of parameter No.387, the program lock is open, otherwise is locked.
[Note] when the value of parameter No.387 is 0, the program lock is invalid, no matter NC power supply is connected or off. Don't put the value other than 0 into parameter if program lock is not needed.	

Other parameters:

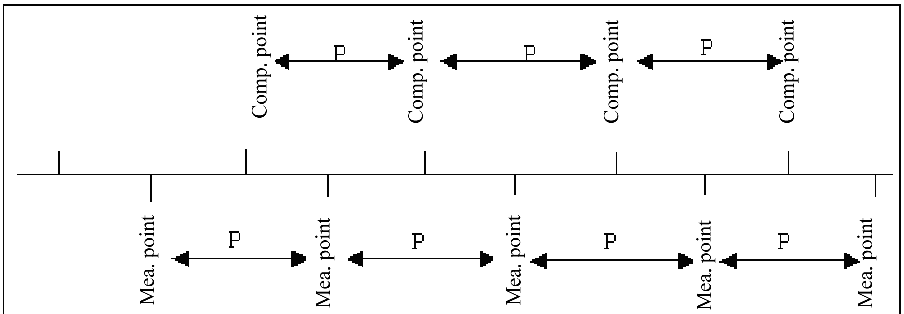
Parameter No.	
1000~ 1127	X axis pitch error compensation setting, setting range: 0~±7 unit:0.001mm
2000~ 2127	Z axis pitch error compensation setting, setting range: 0~±7 unit:0.001mm
Description	

Consult the following

[Note1] Pitch error compensations are active at the end of return to reference point by all axes. Compensation is inactive even parameter for pitch error compensation is set when reference return is not performed. Parameter setting shall be made before reference point return. The actual compensation is got by the parameter setting value multiplying compensation coefficient (see parameter No.024).

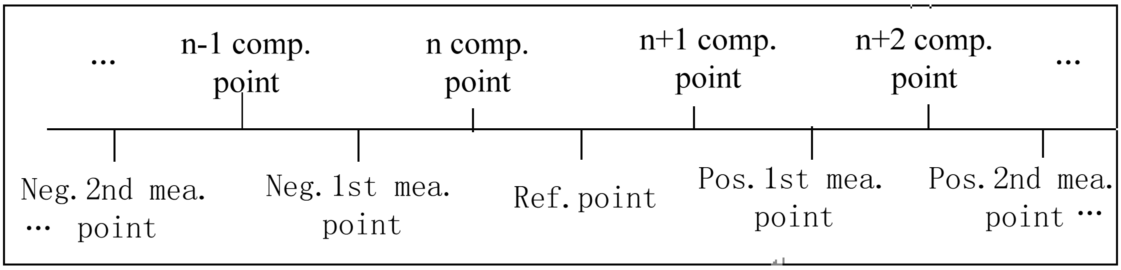
[Note2] As PRM316 ACMR=1 (any commanded detection override), the unit of the pitch error compensation is detection unit.

The point intervals for pitch error are identical that each axis may be set for 128 compensation points (0~127).



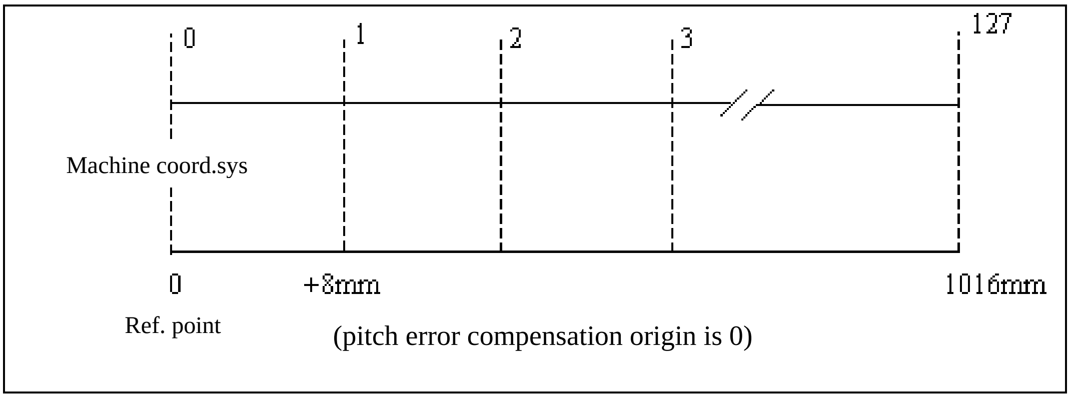
P: Pitch error compensation interval

The min. interval for pitch error compensation is as following:

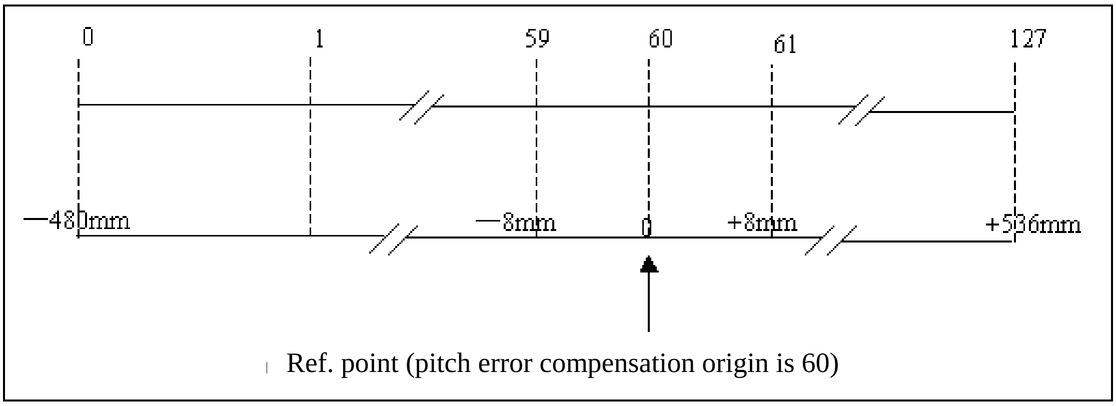


When the pitch error is measured at the measuring points (by parameter No.383, 384), the intervals between measuring points are identical. Set the 1st measuring point error from reference point positive direction to the (n+1) compensation point, and the 2nd , 3rd… measuring point errors to (n+2), (n+3) …compensation points respectively. The compensation method for negative direction is the same.

For example: pitch error compensation origin is 0, the interval is 8.000mm, the 1st set position is +8.000mm, the 127th set position is +1016.000mm, so the compensation is made in a range from 0 to +1016.000mm.



If pitch error compensation origin is 60, the 61st set position is +8mm, the 0 set position is -480mm, the 127th set position is +536 mm, so the compensation is made in a range from -480mm to +536mm.



Pitch error compensation origin is defined by machine stroke and reference point, and it is set by parameter No.336 and 337

Pitch error compensation is set by an absolute (incremental) value, and "±"signs indicate compensation direction.

When an axis move positively to compensate: - error for + compensation: + error for – compensation.

When an axis move negatively to compensate: - error for – compensation: + error for + compensation.

Parameter No.024: compensation coefficient (×1,×2,×3,×8)

336: For X axis pitch error compensation origin setting

337: For Z axis pitch error compensation origin setting

383, 384: For axis pitch error compensation interval setting

1000~1127: For X axis pitch error offset setting

2000~2127: For Z axis pitch error offset setting

Note: offset sign changes according to the move direction when compensation is performing by offset and irrelevant to origin.

Setting data (some user common parameters may also be set in "SETTING")

Data No.	Content
057*	Running time(hour) (TMHOR)
058*	Running time(minute) (TMMIN)
059*	Running time(second) (TMSEC)
064*	Chamfering width of thread (THDCH)
065*	Retraction amount of G74,G75 (GROVE)
066*	Finishing allowance of G76 (THDFN)
067*	Retraction amount of G71,G72 (MRCDT)
068*	Min. cutting depth of G76 (THCLM)
141*	Running time (TIME1)
151*	1st acme X coordinate of memory travel limit 2
152*	1st acme Z coordinate of memory travel limit 2
153*	2nd acme X coordinate of memory travel limit 2
154*	2nd acme Z coordinate of memory travel limit 2
155*	1st acme X coordinate of storage travel limit 3
156*	1st acme Z coordinate of storage travel limit 3
157*	2nd acme X coordinate of storage travel limit 3
158*	2nd acme Z coordinate of storage travel limit 3
180*	Sequence No. to stop at
319*	Various settings (PRGS,MSBL)
340*	For input device selection in storing programs
341*	For output device selection in outputting

·They are set by functional keys.

·Data numbers other than above is displayed blank.

·Data numbers with * sign may also be set in a same data number by PARAM soft key, see the same data number of parameter in the parameter explanation.

Appendix 6 Alarm List

No.	Descriptions	Remarks
000	Cut off power at once after parameter set, then power on (parameter No.000~005,012～015, 018, 027, 028, 031, 032, 082, 086, 087, 090, 124, 125, 128, 129, 300~ 304, 316)	
001	TH alarm(parity error characters have been input in valid msg area)	
002	Error happens in TV check; set the “TV CHECK” to 0 to avoid system TV check	
003	*The data input exceeds the bits allowable; when it is the program name exceeds then move the cursor and press EOB to delete	
004	Digits, character “—” or decimal point were input when the beginning of the block is without address. (For user macro program, see the P/S alarm explanation in Chapter 3 Section 10.11)	
005	There’s no data after address but another address or (EOB) code is input next.	
006	“—”sign is illegal (“—” is input after the address with “—” unallowable or 2 or more “—” are input)	
007	Illegal decimal point (“.” is input after the address with “.” unallowable or 2 or more “.” are input)	
009	Illegal code is entered in the active information area.(B,C,Y,V,J,R)	
010	Illegal G code is commanded (also for the entered G code with no function)	
011	No feedrate or improper feedrate is commanded.	
012	E code commanded with no E6 bits selection	
014	Lead increase/decrease by address K is over the max. Value or negative in variable lead threading.	
022	R is commanded without radius R programming and the first bit of parameter No. 0 is changed to 1.	
023	Radius R is 0 or negative in arc compensation commanded by R.	
029	Offset value has exceeded 6 bits. Reset.	
030	Tool position offset is too large.	
031	P value which specifies offset number is too large or without address in offset input (by G10 or user macro input command)	
032	The offset is too large in the offset inputting (by G10 or user macro program)	
033	The intersectional point can’t be got by tool nose R compensation C. Or tool nose compensation B specified less than 90° intersection point compute. Modify the program or set the 4th and 3rd bits of the parameter No.1 to 0 and 1 respectively.	
034	Compensation “start” or “cancellation” is made in G02/G03 in tool nose R compensation. Modify the program.	
035	Skipping cutting (G31) is commanded in tool nose R compensation.	
038	Overcutting occurs due to the arc start or end point coincides with the center in tool nose R compensation.	

039	Overcutting occurs due to the chamfering, round corner R at start, cancellation or G41/G42 switching in tool nose R compensation.	
040	Overcutting occurs in tool nose R compensation of canned cycle G90/G94.	
041	Overcutting occurs in tool nose R compensation.	
047	G27 to G30 are commanded for an axis without reference point return. Modify the program or set the 6th and 7th bits of parameter No.0 to 1, the 0~3rd bits of the parameter No.20 to 1.	
048	1. G30 is commanded without returning to reference point after power on or emergency stop. 2. When a storage type travel limit option is provided, a move is executed without returning to reference point after power on or emergency stop.	
050	Chamfering and corner R are specified in a thread cutting block.	
051	No G01 command in next block after chamfering and corner R block.	
052	Move direction or amount is not right in a block after chamfering and corner R block.	
053	2 or more I, K, R in a block for chamfering and corner R command	
054	Taper command in chamfering and corner R block	
055	Move amount less than chamfering and corner R in chamfering and corner R block.	
056	Address X (or Z) and I (or K) commanded in chamfering block. G01 X(U) I; or G01 Z(W) K	
059	The program of the selected part number is not found (external part number selection A function).	
060	Sequence number specified is not found in the searching or program restart.	
061	Address P or Q is not specified in the block that has G70, G71, G72, G73.	
062	·Value behind address D is not integer in the block that has G71, G72, and G73. ·Values specified by K, D are not positive or by A is a disabled angle in the block that has G76. ·Values specified by I, K, D are negative in the block that has G74, G75. ·X is commanded whether I value is 0 or not, or Z is commanded whether K value is 0 or not in the block that has G74, G75.	
063	Sequence number specified by P, Q is not found in the block that has G70, G71, G72, and G73.	
065	G00 or G01 of group 01 is not commanded for G71, G72, and G73 in a block that is numbered by P. Z(W),(G71) or X(U)(G72) is commanded for G71,G72 in a block that is numbered by P.	
066	G code other than G00,G01,G02,G03,G04 is commanded for G71,G72,G73 between blocks specified by P and Q.	
067	G70, G71, G72, G73 are specified in MDI mode.(they are only for Auto mode)	
068	The concaves are over 10 for G71, G72 of II type.	
069	The last move command of the block specified by P and Q is ended by	

	chamfering or corner R when command with G70, G71, G72, and G73.	
070	There's no space in memory.	
071	The address searched is not found.	
072	The number of programs stored exceeds 95 or 191.	
073	Program number is already used.	
074	Program number is beyond the range of 1～9999.	
075	Neither program number nor sequence number is entered in the first block of the program.	
076	Address P is not commanded in the blocks that containing M98, G65, G66.	
077	Subprogram is called for 3 times (5 times for the user macro program)	
078	Program number(by G,M,T) or sequence number specified by address P or the sequence number in GOTO statement is not found in the blocks which includes M98, M99, G65, G66.	
080	Measuring position in-position signal is not connected within the area by parameter ε.(automatic tool compensation)	
081	Automatic tool compensation without T code is commanded. (automatic tool compensation)	
082	Automatic tool compensation and T code are commanded in a same block. (automatic tool compensation)	
083	Inactive axis or incremental command is specified in automatic tool compensation (automatic tool compensation).	
088	Data error signal (DERR) is entered.(external tool compensation A function)	
089	Value other than 0～9 is used for BCD data. (external tool compensation A function)	
090	Reference return can't be normally performed due to the one-turn signal (the reference point signal for linear scale)from the pulse encoder is not entered during its returning in grid mode.	
091	Reference return can't be normally performed due to the asynchronization between one turn signal (the basic point signal in linear scale) feedback from the pulse encoder and the reference counter for too low speed during its returning in grid mode.	
092	The axis commanded by G27 is not returned to the reference point.	
094	P type command cannot be specified for program restart (the coordinate system setting and origin setting operations etc. are done after the program interruption.)	
096	P type cannot be specified for the program restart. (workpiece coordinate system offset changed by program interruption)	
097	P type cannot be specified for the program restart. (auto running is not executed after power on, the releasing of emergency stop and overtravel alarm (instant stop))	
098	G28 is found in searching without reference point return by program restart command after power on or the releasing of emergency stop and overtravel alarm (instant stop).	
099	Move is commanded in MDI mode after the program restart check.	

100	Parameter write switch is ON, RESET key is pressed after this switch is set for OFF.	
101	In the part program editing and storing for memory writing, the power is cut off. As the alarm occurring, press DELETE and RESET key simultaneously for power-on to clear the memory.	
110	The absolute values of data by decimal point are beyond the range allowable.	
111	Datum exponent by floating point exceeds the range.	
112	Divisor is zero.	
113	Function unallowable for user macro program A is used.	
114	Format other than < expression> is wrong.	
115	Value that can't be defined by variable is commanded.	
116	The left of the expression is the impermissible variable.	
118	Parentheses nesting are beyond upper limit (5).	
119	The argument of SQRT is negative; or that of the BCD is negative.	
122	The calling times of macro program are beyond the range (1~4).	
124	DO-END does not correspond one by one.	
125	The format of FORMULA is wrong.	
126	Value n is beyond the range $1 \le n \le 3$ in DO_n statement.	
127	Commands of NC, MACRO are mixed together.	
128	Value n is beyond the range $0 \le n \le 9999$ in $GOTO_n$ statement.	
129	Address unallowable is used in <argument assignment>.	
130	The bigger address data of external data is incorrect.	
131	5 or more alarms occur in external alarm message.	
132	Alarm numbers are not found in the canceling of the external alarm message.	
133	Data of small addresses data are wrong in the external alarm message or the external operation message.	
134	The tool group numbers are over range (16).	
140	Tool no. used is larger than 9932.	
141	The tool group commanded in part program is not set.	
142	The tools in a group are over 16.	
143	T code is not stored in the block of a program for tool group setting.	
145	The tool group code △△ by T△△88 does not match the relevant T code in service.	
146	There are no P and L commands at the beginning of the program for tool group setting.	
147	The tool groups are over 16.	
150	Corner R is commanded in the block of threading or rapid command.	Dialogue programming
151	Dimension direct input is commanded in the block containing the following command: (dialogue programming) 1） Non-modal G code other than G04 2） G02, G03, G90, G92, G94 or either of them in 01 group	Dialogue programming

152	Chamfering or corner R is not allowed for insertion. (too large.)	Dialogue programming
153	The address behind (,) is not C or R in drawing dimension direct input.	Dialogue programming
154	Drawing dimension direct input is commanded for cutting direction block in compound canned cycle (G71, G72) Ⅱ type.	Dialogue programming
155	No move is commanded in the block that has chamfering or corner R or in next block.	Dialogue programming
156	2 lines intersection can't be defined by 2 blocks containing drawing dimension direct input.	Dialogue programming
157	In block X(U)_ _A_ _; angle command (A) depart from the range of Z axis (0° or 180°) ±1°, in block Z(W)_ __A__ __; angle command A depart from the range of X axis (90° or 270°) ±1°.	
158	Intersection can't be got for two lines angular difference is within ±1° in drawing dimension direct input.	
170	Alarm is issued for No. 8000～8999, 9000～9899 program edit which is disabled by parameters. (See parameter No. 318－PRG9, No.319－PRG8)	
210	It contacts the positive travel limit of X axis.	
211	It contacts the negative travel limit of X axis.	
212	Tool enters the exclusion area of storage type travel limit 1 when X axis moves in the positive direction.	
213	Tool enters the exclusion area of storage type travel limit 1 when X axis moves in the negative direction.	
214	Tool enters the exclusion areas of storage type travel limit 2, 3 when X axis moves in the positive direction.	
215	Tool enters the exclusion areas of storage type travel limit 2, 3 when X axis moves in the negative direction.	
220	It contacts the positive travel limit of Z axis.	
221	It contacts the negative travel limit of Z axis.	
222	Tool enters the exclusion area of storage type travel limit 1 when Z axis moves in the positive direction.	
223	Tool enters the exclusion area of storage type travel limit 1 when Z axis moves in the negative direction.	
224	Tool enters the forbidden areas of storage type travel limit 2, 3 when Z axis moves in the positive direction.	
225	Tool enters the forbidden areas of storage type travel limit 2, 3 when Z axis moves in the negative direction.	
401	No drives ready signals of X axis and Z axis arriving CNC. Check the drive alarm number or the connection.	
401.1	No drives ready signal of X axis arriving CNC. Check the drive alarm number or the connection.	
401.3	No drives ready signal of Z axis arriving CNC. Check the drive alarm number or the connection.	
404	Even though the ready signal of position control (PRDY) has been cut off,	

	the ready signal of the speed control (VRDY) is not cut off. Or if the power is switched on, the VRDY signal has not been set for ON before PRDY signal does.	
405	Reference point return can't be performed due to the fault of the NC interior and servo system. Use manual reference return.	
406	There is fault in position detection system when doing servo feeding feedback, and axes single alarm 414, 424 are also given together.	
410	The position offset at the stop of X axis exceeds the setting.	
411	The position offset in the moving of X axis exceeds the setting.	
412	X axis offsets excessively (above 500VELO).	
413	The position offset of X axis exceeds ±32767 or the speed command value of DA converter goes beyond the range from +8191 to -8192. The alarm is usually a result of incorrect settings.	
414	Abnormity occurs in the detection device of rotary transformer and inductosyn scale of X axis	
415	A speed above 511875 detection unit/s is commanded for X axis. The alarm is the result of incorrect CMR setting.	
416	The position detection device for the pulse encoder of X axis is out of order (disconnection alarm).	
417	The servo position loop LSI of X axis is incorrect.	
420	Position offset exceeds the setting when Z axis stops.	
421	Position offset exceeds the setting when Z axis moves.	
422	Z axis drifts excessively (more than 500VELO).	
423	The position offset of Z axis exceeds ±32767 or the speed command value of DA converter goes beyond the range from +8191 to -8192. The alarm is usually a result of incorrect settings.	
424	Abnormity occurs in the detection device of rotary transformer and inductosyn scale of Z axis.	
425	A speed above 511875 detection unit/s is commanded in Z axis. The alarm is the result of incorrect CMR setting.	
426	The position detection device for the pulse encoder of Z axis is out of order. (disconnection alarm)	
427	The servo position loop LSI of Z axis is bad.	
600	Data transfer of the connection units is wrong.	
603	The communication between NC and PC is improper or disconnected.	
604	The CPU of PMC-Ⅱ cannot be effectively held on.	
605	System alarm is given in the CPU of PMC-Ⅱ. (monitor alarm)	
606	RAM/ROM parity error occurs in the CPU of PMC-Ⅱ.	
607	The data transfer of MDI/LCD is wrong.	
920	System error (monitoring timer alarm).	NC alarm
930	CPU error (0, 3 and 4 type interruption occurring)	NC alarm
940	Offset memory alarm (excessive offset is set): Set correct offset in a specified offset number.	NC alarm
960	The temporary storage area for system control commands becomes inadequate.	NC alarm

Appendix 7 State List of Power On Reset&Clearing

○: A state remains unchanged or action is kept on. ×: A state is cancelled or an action is interrupted.

	Item	At power on	Clearing state	Reset state
Setting data	Offset	○	○	○
	Setting data	○	○	○
	Parameter	○	○	○
Other data	Programs in memory	○	○	○
	Information in buffer	×	×	○ (In MDI mode) × (Other modes)
	Display of sequence numbers	×	○ (Note 1)	○ (Note 2)
	Non-modal G codes	×	×	×
	Modal G codes	Initial value (G20/G21 remains constant as before power off)	Initial values, but G20/G21 and G22/G23 remain unchanged.	All remain unchanged.
	F, E	Zero	Zero	○
	S, T, M	×	○	○
	L	×	×	○ (In MDI mode) × (Other modes)
Coordinate	Workpiece coordinates	Zero	○	○
Executing motions	Move	×	×	×
	Dwell	×	×	×
	M, S, T, B codes are sent.	×	×	×
	Tool length compensation	×	Set by the 3rd bit "RS43" of parameter No.22"	MDI mode: ○ Other modes: set by the 3rd bit "RS43" of parameter No.22"
	Tool radius compensation	×	×	○ (In MDI mode) × (Other modes)
	Saving of the subprogram numbers called	×	× (Note 2)	○ (In MDI mode) × (Other modes) (Note 2)

Indicator and output signals	ALM	The indicator goes out in the absence of alarm.	The indicator goes out in the absence of alarm.	The indicator goes out in the absence of alarm.
	NOT READY	×	×(light is on in emergency)	×(light is on in emergency)
	LSK	×	×	○ (In MDI mode) × (Other modes)
	BUF	×	×	○ (In MDI mode) × (Other modes)
	Reference return		○(for emergency stop×)	○(for emergency stop×)
	S·T·B code	×	○	○
	Spindle rotation signal(S12bit/S analog signal)	○	○	○
	NC ready signal (MA,MB)	ON	○	○
	Servo ready signal	ON(no servo alarm)	ON(no servo alarm)	ON(no servo alarm)
	Indicator for cycle start	×	×	×
	Indicator for feed hold	×	×	×

Note 1: Program number is displayed when the program is starting from the head.

Note 2: When the NC is reset in the execution of a subprogram, the control returns to the beginning of the main program. And the called subprogram cannot be executed in midway. So the control is back to the program head**.**

Appendix 8　Storage Type Pitch Error Compensation Function

H8.1　Function

The ball screw error of all axes may be compensated by minimum move units. This function is active after reference point returning.

H8.2　Specifications

The tool position after returning to reference point is taken as the reference. The compensation origin (parameter No. 336, 337), interval (parameter No. 383, 384), amount (parameter No. 1000～2127) of each axis pitch error may be set by parameters. And the compensation amount may be multiplied by 1, 2, 4 and 8 according to the pitch error compensation override (parameter No.024).

(Ⅰ) Compensable axes: X, Z axes

(Ⅱ) Compensation points:128 (127 intervals)

(Ⅲ) Compensation value

Compensation point: 0～±7×compensation override(minimum move unit)

Compensation override　×1, ×2, ×4, ×8 (for all axes)

(Ⅳ) Compensation interval:

Move unit	Minimum set interval	Maximum set interval	Unit
Metric system	8, 000	20, 000, 000	0.001 mm
Inch system	4, 000	20, 000, 000	0.00001 inch

(Maximum compensation distance = set interval ×128)

Actual compensation interval shall be set depending on the optimal value between the maximum compensation distance and machine travel in the ranges as listed in the above table.

When the set interval is less than the above minimum set interval, compensation is impossible at a predetermined position. Now it is necessary to reduce the feedrate.

The minimum set interval of X axis by diameter is

16000 mm (Metric system)

8000 inch (Inch system)

Here the pitch error compensation data is set by diameter.

H8.3　Parameter

Parameters concerning pitch error shall be set in the following parameter numbers in MDI mode or emergency stop mode.

H8.3.1　Pitch error compensation override

0	2	4

	PML_2	PLM_1						
Parameter No.	7	6	5	4	3	2	1	0

PLM2, 1: The override is multiplied by the compensation set by parameter(＃1000～2127) then output.

PLM_2	PLM_1	Override
0	0	×1
0	1	×2
1	0	×4
1	1	×8

(for all axes)

H8.3.2 Pitch error compensation origin

3	3	6	PECZRX
3	3	7	PECZRZ

PECZRX, Z: Pitch error origins for X, Z axes

They are used to set origins of pitch error compensation list.

Set the value for the range 0 ~ 127 according to axes of machine.

H8.3.3 Pitch error compensation interval

3	8	3	PECINTX

3	8	4	PECINTZ

PECINTX, Z: Pitch error compensation intervals for X, Z axes

They are used to set intervals of pitch error compensation

They are usually set for 8000 or above (Metric), or 4000 or above (Inch)

And the setting of 0 is not compensated for axis.

Setting unit: 0.001 mm (metric)

0.0001 inch (inch)

The setting of X axis should be above 16000 (metric) or above 8000(inch) if X axis is programmed by diameter. And the parameter should be set by diameter too.

.

H8.3.4 Pitch error compensation value

The pitch error compensation values for all axes are set in the following parameters:

Axis	Parameter No.
X	1000～1127
Z	2000～2127

Compensation cannot be set for the parameter numbers other than the listed in the above table. The set compensation range is 0～±7. The setting beyond the range is inactive.

Setting unit: 0.001 mm (metric)

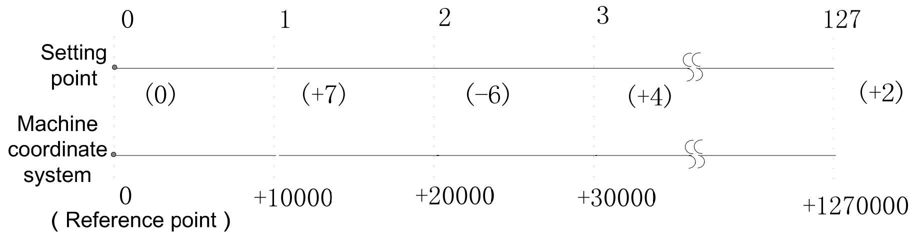
0.0001 inch (inch)

Use diameter if X axis is commanded by diameter.

This value can be multiplied by 1, 2, 4, 8 by parameter No.＃24.

H8.4 Examples of Parameter Setting

(1) Pitch error origin =0, compensation interval =10000



The beginning of the compensation list corresponds to the reference point while compensation point 1 corresponds to the point by 10000 from the reference point in the positive direction. After that, every 10000 point corresponds to a compensation point. So the compensation point 128 is set at 1270000. i.e. The compensation for the travel from 0 to 10000 is set at compensation point 1 while that from 10000 to 20000 is set at compensation point 2 and that from (n－1)×(compensation interval) corresponds to n×(compensation interval) is set at compensation point n.

In the above example, the zone

0～10000 set to －7 (Compensation value)

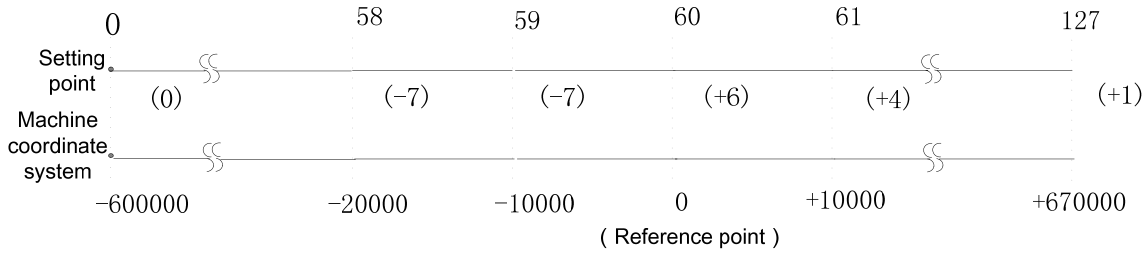
10000～20000 set to＋6 (Compensation value)

20000～30000 set to－4 (Compensation value)

When the machine moves from the reference point to the position at +30000, the total compensation is as follows:

(＋7)＋(－6)＋(＋4)＝＋5

(2) Pitch error origin =60, compensation interval =10000



The No.61 (compensation point 60) of the list corresponds to the reference point while compensation point 61 corresponds to the point by 10000 in the positive direction from the origin. After that, every 10000 point corresponds to a compensation point. So the compensation point 128 is set at 670000. In addition, compensation point 59 corresponds to the point by 10000 in the negative direction from the reference point. i.e. a compensation value from (n-60-1)×(compensation interval) to (n-60)×(compensation interval) is set at compensation point n.

In the above example, a zone compensation data is as following:

-20000～10000 set to +7 (Compensation value)

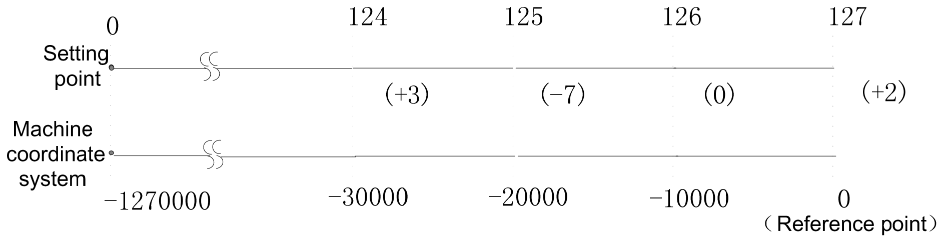
-10000～0 set to -6 (Compensation value)

0～+10000 set to －4 (Compensation value)

When the machine moves from -20000 to +10000, the total compensation is as follows:

(－7)＋(＋6)＋(＋4)＝＋3

(3) Pitch error origin =127, compensation interval =10000



The end of the compensation list corresponds to the reference point while compensation point 126 corresponds to the point by 10000 in the negative direction from the reference point. After that, every 10000 point corresponds to a compensation point. So the compensation point 0 is set at -1270000. i.e. The compensation for the travel from -10000 to 0 is set at compensation point 127, while that from -20000 to -10000 is set at compensation point 126, and that from (n－128)×(compensation interval) to (n-127)×(compensation interval) is set at compensation point n.

In the above example, a zone compensation data is as following:

－40000～－30000 set to －3(Compensation value)

－30000～－20000 set to ＋7(Compensation value)

－20000～－10000 set to 0(Compensation value)

－100000～0 set to －2(Compensation value)

When the machine moves from -40000 to the origin, the total compensation is as follows:

(＋3)＋(－7)＋(0)＋(＋2)＝－2

H8.5 Compensation value determination

For pitch error compensation,

·Distance from reference point to origin

·Machine move direction

·Pitch error compensation interval

The set compensation value has no direct relations with above factors. The compensation at the compensation point n (n=0, 1, 2……127) is only defined by the zone mechanical error of (n-compensation origin -1)×(compensation interval) ～ (n－compensation origin)×(compensation interval). i.e. compared the absolute value of actual displacement amount of mechanism, setting for the compensation positions is as follows:

＋error(move more), set to －compensation value

－error(move less), set to ＋compensation value

H8.6 Others

H8.6.1 Cancellation of pitch error compensation

In pitch error compensation setting, if a previous value of an axis is required to be eliminated, it can be done by following methods:

(1) Select the parameter No. which set the axis compensation to be eliminated;

(2) Set the compensation to -9999.

Either of the selected parameter No. in (1) for compensation of the axis is available for elimination.

H8.6.2 Precautions for setting

(Ⅰ) Compensation interval setting (parameter No.383, 384)

As interval is:

positive, compensated by positive value;

negative, compensated by its absolute value;

Zero, not compensated.

(It is shown by an absolute one even a negative compensation interval is input)

(Ⅱ) Pitch error compensation is active after power off and restart and at the end of reference point return. Compensation is not made even though parameter setting if reference point return is not performed.

Furthermore, parameter setting should be made before reference point return after power on. If pitch error compensation parameter is to be changed after reference point return, the reference point should be returned again and power off and restart.

(Ⅲ) Pitch error compensation value (parameters No. 1000 ～ 2127)

The following restrictions are made for pitch error compensation:

The value of (compensation value of pitch error)×(pitch error compensation override)×CMR must fall within ±127.

If the setting of the value exceeds ±127, compensation cannot be correctly made. If it is necessary to set a value beyond ±127, divide the compensation and then make compensation at adjacent compensation points.

Note: CMR: Command override multiplier, refer to parameters No.27, 28.

Appendix 9　Operation List

Classification	Function	Program protection switch	Parameter write switch	Mode	LCD	Operating procedures
Clear	To clear memory		ON	At power ON	—	Concurrently press the O and DELETE keys to switch on the power.
	To clear parameters		ON	At power ON	—	Concurrently press the CANCEL and DELETE keys, or PARAMETER and DELETE keys to switch on the power.
	To clear programs			At power ON	—	Concurrently press the RESET and DELETE keys, or PROGRAM and DELETE keys to switch on the power.
MDI input	To input parameters		ON		PARAMETER	N → Parameter No.→ INPUT → P →data → INPUT → parameter write switch OFF → RESET
	To input offset	OFF		Any	OFFSET	N → offset No. → INPUT → * → offset data → INPUT
						(Note:*　stands for X, Z, R, T, U or W)
	To input data	OFF		MDI	SETTING	Move the cursor to the item to be altered → P →data → INPUT

Search	Program number search			Edit	PROGRAM	(1) O →Program number →↓ (cursor) (2) O →O → ↓ (cursor)
				Auto	PROGRAM	(1) O →Program number → ↓(cursor) (2) O →CANCEL→ ↓(cursor)
	Sequence number search			Auto Edit	PROGRAM	Program number search →N→sequence number → ↓ (cursor)
	word search			Edit	PROGRAM	Input the address or data to be searched →↓ (cursor)
	Address search			Edit	PROGRAM	Input the address to be searched→↓ (cursor)

Classification	**Function**	**Program protection switch**	**Parameter write switch**	**Mode**	**LCD**	**Operating procedures**
Program	To delete all	ON		Edit	PROGRAM	O → 9 9 9 9 → DELETE
	To delete one	ON		Edit	PROGRAM	O → program number→ DELETE
	To delete blocks	ON		Edit	PROGRAM	N →sequence number→ DELETE
Edit	To delete a block	ON		Edit	PROGRAM	Search the block to be deleted → EOB → DELETE
	To delete a word	ON		Edit	PROGRAM	Search the word to be deleted→ DELETE
	To alter a word	ON		Edit	PROGRAM	Search the word to be altered →address → data → ALTER
	To insert a word	ON		Edit	PROGRAM	Search the word preceding the one to be inserted →address →data → INSERT

	Memory sorting	ON		Edit	PROGRAM	CANCEL → SHIFT
RS232 data input output	All prg output			Edit	PROGRAM	O → 9 9 9 9 → DATA OUTPUT
	A prg output			Edit	PROGRAM	O → program number → DATA OUTPUT
	Program input	OFF		Edit	—	File name → O → - 9 9 9 9 → DATA INPUT
	Offset data output			Edit	OFFSET	P → - 9 9 9 9 → DATA OUTPUT
	Offset data input			Edit	—	File name → O → program number → DATA INPUT → program execution
	Parameter output			Edit	PARAMETER	P → - 9 9 9 9 → DATA OUTPUT
	Parameter input		ON	Emergency ON	PARAMETER	File exported → P → - 9 9 9 9 → DATA INPUT

Appendix 10　Program Lock

J10.1　General

Program numbers 9000～9899 can be locked by parameters. In lock mode, the programs No. 9000 ～ 9899 cannot be displayed, edited and output. The function may be used to protect the special programs developed by user macro programs and to prevent them from accidental deletion.

J10.2　Program Number

It is possible to lock the programs No. 9000 ～No. 9899 with keys. Other programs cannot be locked with keys. Once locked with keys, all the programs of No. 9000 ～ No. 9899 are automatically locked. Therefore, the programs that do not need locked should use the numbers other than 9000 ～ 9899.

J10.3　The State After Key Locking

In key locked mode (see Section J10.4 below), the state of the programs No. 9000～ 9899 are as follows:

(1) Their information is not displayed even in execution.

(2) Program number search (alarm No.071) is impossible in Edit mode (Auto mode). Therefore they cannot be edited.

(3) The memory cannot be sorted.

(4) Their numbers are not included in the display of all program numbers.

(5) Program output is impossible (not output even all programs are output).

(6) Program deletion is impossible (not deleted even all programs are deleted).

(7) Program storage is impossible (alarm No.170).

J10.4　Key Lock and Unlock Procedures

(1) Preset a password (1 to 99999999) in parameter No.387. Take down the password since the contents of the parameter is not displayed. Programs cannot be locked if it is set to 0.

(Note 1)　The setting of the parameter is only active in unlock state.

(Note 2)　The parameter is not cancelled even in parameter complete clearing state.

(Note 3)　The parameter turns 0 after memory's clearing completely, i.e. the program lock is released.

(2) To enable the unlock, set the same value in parameter No.408 as that in parameter No.387.

The unlock is enabled only when it has the same value as in parameter No.387.

The settings of the parameter are not displayed.

(Note 1)　The parameter cannot be stored in SRAM memory.

(3) Method of key lock after unlocking:

（a）Set a different value in parameter No.408 with that in No.387.

（b）Switch off the NC power once and then switch on it again.

Parameter No.

3	8	7	Password number

Store the password number for lock.

Setting range: 1～99999999

4	0	8	Lock/unlock

The program lock is disabled by entering the same value as in parameter No.387.

The program lock is active when a different value is entered.

Note 1: When the value of parameter No.387 is set to 0, the program is unlocked and remains unlocked even the NC power is switched ON/OFF.

Note 2: Don't set a value other than 0 in parameter No.387 when lock is not necessary.

J10.5 Cautions

(1) Proceed as follows if the set password (No.387)is forgotten:

(a) Completely clear the memory (unlock).

(b) Input all parameters (except No.387)

(c) Save program number 9000~9899 into memory.

(d) Set a password in parameter No.387 (lock).

(2) After O9000~O9899 programs saved and edited, other programs beyond O9000~O9899 should be called before the lock function is enable. When the program number is 9000～9899, it is time to set the password and switch to Edit mode, and finally enter into PROGRAM interface as the following shows. This interface is disappeared when new program beyond O9000~O9899 is created..

```
PROGRAM                         O9080 N0801
***CAUTION***
   STOP   PUSH    RESET
   CONTINUE: RETURN MODE TO
             MEMORY OR TAPE

                          LSK  BUF  INC
```

Appendix 11 Interruption Function of User Macro Program

K11.1 General

During the execution of a program, it is possible to call another program by inputting interruption signal at the machine side. This is called interruption function of user macro program.

The interruption command in a program is as follows:

M96 P××××; user macro program interruption ON

M97; user macro program interruption OFF

By using this function it is possible to call another program on any program block being executed and start program operations in ever-changing conditions.

(Example of application)

(1) Start abnormal tool detection with external signal.

(2) Stop the currently performing machining and insert other machining in a series machining.

(3) The current machining information is read regularly.

It is also applicable for the adaptive control.

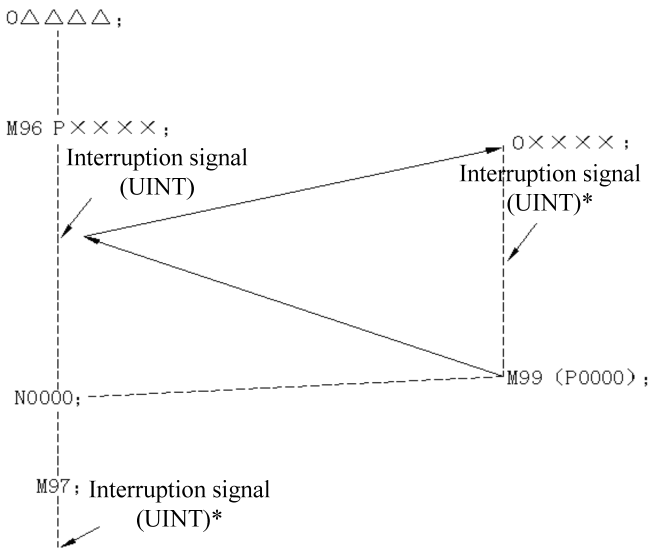


Fig. 1 user macro program interruption

After M96P×××× is commanded in program, the currently executing program is interrupted to execute the program commanded by P×××× once interruption signal (UINT) is input.

The following interruption signal(UINU) during the execution of an interruption program and after M97 is inactive (marked "*" in the figure).

K11.2 Command Methods

K11.2.1 Significant conditions

User macro program interruption is only effective during the execution of a program. Namely the significant conditions are:

(1) Auto mode is selected.

(2) STL (start light) is set to ON.

(3) User macro program interruption is yet not being executed.

User macro program interruption cannot be executed in manual operation (JOG, STEP, HANDLE etc).

K11.2.2 Format of command

The interruption signal (UINT) of user macro program interruption function is set to be disabled or enabled by M96 and M97 codes in principle.

That is, when M96 is commanded, interruption signal (UINT) may be used to start user macro program interruption until M97 is commanded or NC is reset. Vice versa user macro program interruption cannot be started even by inputting interruption signal (UINT) after M97 or reset. And interruption signal (UINT) is ignored before M96 is commanded.

(Format)

M96 P××××; User macro program interruption ON

Specifying an interruption program number

M97; User macro program OFF

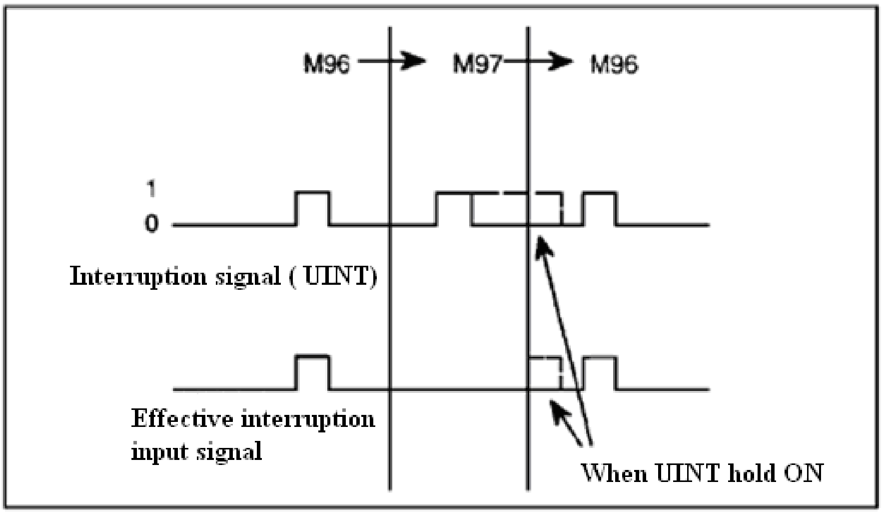


Fig. 2.2 Relation between M96, M97 and interruption signal

Once M97 is commanded, interruption signal will be inactive. The interruption signal input after M97 is kept until the input of M96. Once M96 is commanded, user macro program interruption can be started at once.

K11.3 Fuction specification

K11.3.1 User macro program interruption

ENABLE / MASK

Even user macro program interruption is not used, it is not necessary to change the program. For this purpose, parameter ENABLE/MASK (025——MUSR) is provided for selecting user macro program interruption.

If user macro program interruption is set to MASK in the parameter, M96 and M97 will be output to the outside as common M codes. If it is set to ENABLE, M96 and M97are processed internally without inputting to the outside.

K11.3.2 Subprogram type and macro program type interruptions

The modes of user macro program interruption include subprogram type interruption and macro program type interruption. Therefore, the parameter (025——MSUB) is designed for selecting an interruption mode.

- Subprogram type interruption

  Interruption program is called by a subprogram.

  That is, the values of local variables remain unchanged before and after interruption. In addition, the interruption is not counted as the nesting of calls.

- Macro program type interruption

  Interruption program is called by a macro program.

  That is, the values of local variables change before and after interruption.

  In addition, the interruption is not counted as the nesting of calls.

  The calls nesting of subprograms and macro programs executed in interruption program are accumulated to their numbers respectively.

  Variables cannot be assigned from the executing main program even a user macro program interruption is a macro program type interruption.

K11.3.3 M codes for user macro program interruption

In principle, M96 and M97 are used for the control of user macro program interruption. However, they may be used for other purposes upon manufacturer's requirements (M function, user macro program M code call, etc.). Therefore, whether these M codes are active or not depends on the setting of parameter (025——MPRM).

When the M codes for control of user macro program interruption are set by a parameter:

The M codes for user macro program interruption ON are set in parameter No. #053.

The M codes for user macro program interruption OFF are set in parameter No. #054.

That M code is not set by the parameter is selected for parameter MPRM, M96 and M97 become the M codes for control of user macro program interruption and parameter No. #053 and #054 are not taken into consideration.

In any case, the M codes for control of user macro program interruption are processed internally without output.

It is recommended not to user the M codes other than M96 and M97 for the control of macro program interruption in consideration of the interchangeability of program.

K11.3.4 User macro program interruption and NC statement

User macro program interruption is of two types: interrupting the executing NC command or waiting for the end of the currently executing program. For this purpose, a parameter (314——MINT) is designed for switching between interrupting in the midway/at the end of block.

If interrupting in the midway is selected by the parameter(type Ⅰ):

(1) Once interruption signal (UINT) is input, the executing move or dwell is interrupted and the interruption program is executed.

(2) When there is an NC statement in the interruption program, the interrupted block command disappears and the NC statement in the interruption program is executed.

In case of return, the next block at the interruption of the original program is executed.

If there is no NC statement in the (a) interruption program, the interrupted block is continuously executed when M99 is used to return to the original program. On the other hand it is interrupted (type Ⅱ) at the end of a block:

(1) Once interruption signal (UINT) is input, the interruption program instead of the command of the currently executing block is interrupted.

(2) When there is an NC statement in the interruption program, it is executed at the end of the executing block.

In any case, the control switches to the interruption program once interruption signal is input.

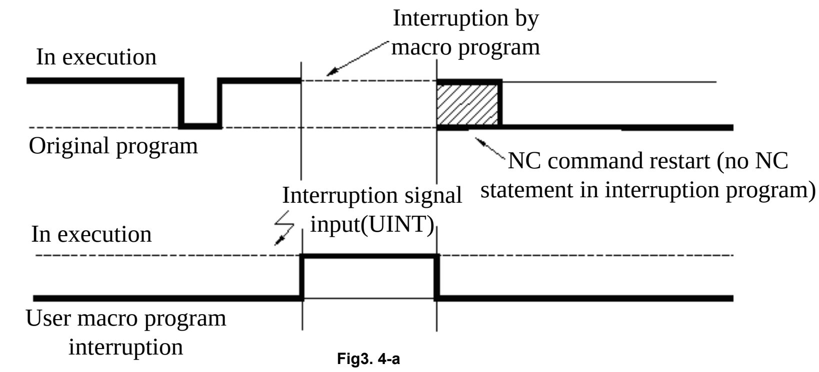


Fig3. 4-a

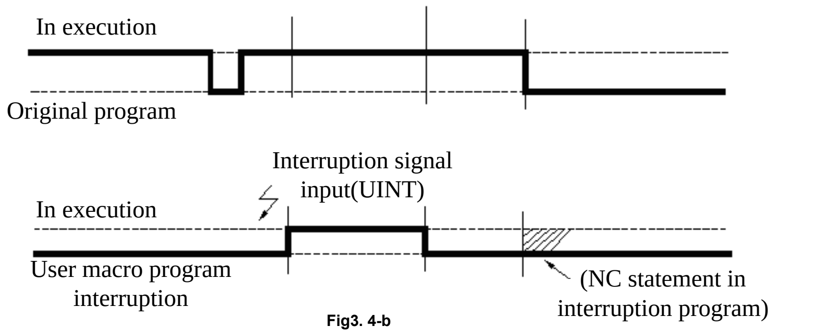


Fig3. 4-b

Fig. 3.4　User macro program interruption and NC statement

The relationship between the interruption in the midway of block (Fig.A-16(a)) and the interruption at the end of block (Fig.A-16(b)) is as above figure. Interruption will be executed wherever interruption signal is input.

K11.3.5　Receiving of user macro program interruption signal (UINT)

There are two modes of receiving of user macro program interruption signal (UINT): state triggering and edge triggering. The so-called state triggering is that it is active in the ON state of signal. Edge triggering is that the rise edge signal is active when the storage signal is switched from OFF to ON.

The use of a mode shall be determined by parameter (025——TSE).

If the parameter is set to state triggering mode, if the interruption signal is active, user macro program interruption can be executed when signal (UINT) is ON, therefore, interruption program can be repeatedly executed when the signal (UINT) is continuously ON.

In addition, when the parameter is set to edge triggering mode, interruption program is completed instantly (only the program commanded by macro program, etc) because it is only active

during the rising edge of the interruption signal (UINT). Therefore, it is only applicable for the occasions that is not suitable for state triggering mode and only one user macro program interruption is performed in the whole program (interruption signal is held ON in this case).

Except special purposes, the actual effects of the two modes are the same (there is no difference in the time from the input of signal to the actual execution of interruption).

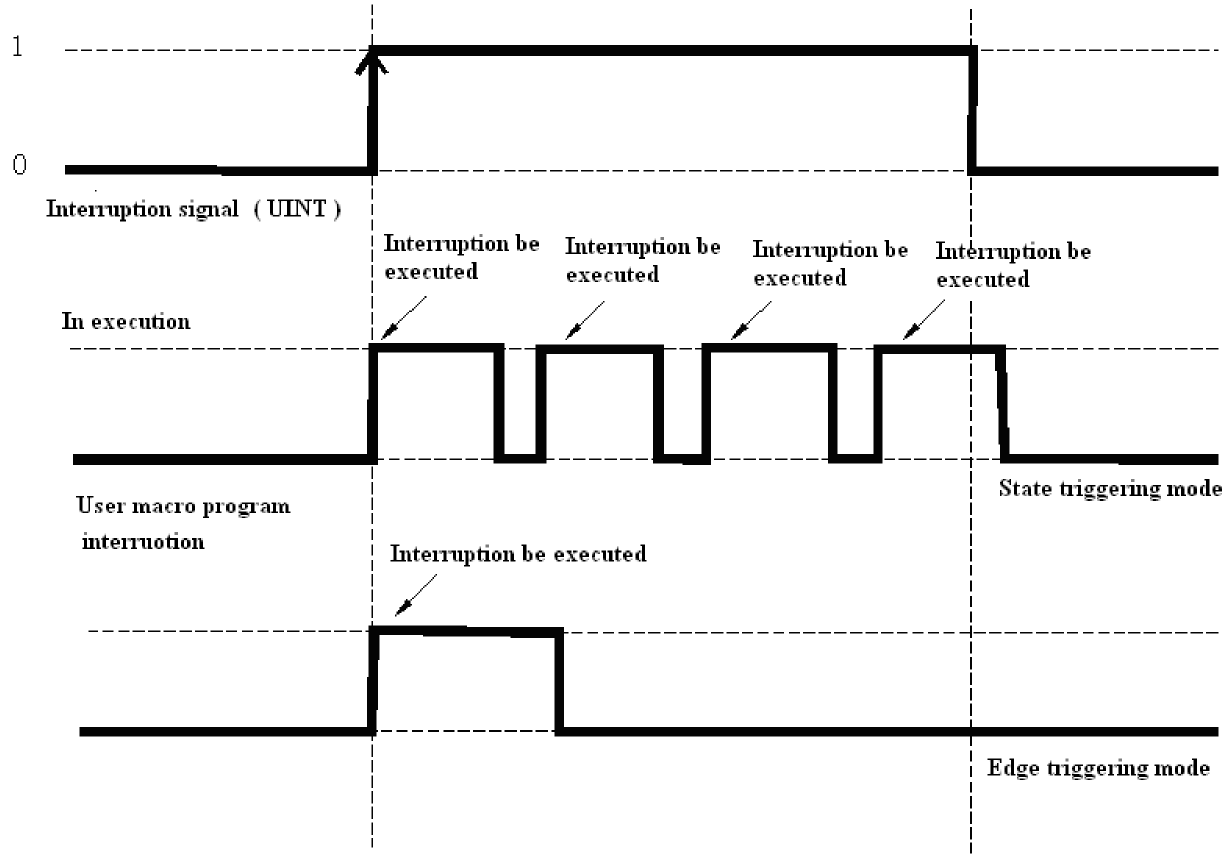


Fig. 3.5 User macro program interruption signal

State triggering performs user macro program interruption in the ON state of signal. Edge triggering performs user macro program interruption in the rise edge of signal. Therefore, in the above example state triggering performs 4 interruptions while edge triggering only one.

K11.3.6 Return from user macro program interruption

M99 is the command for returning from user macro program interruption to the original program. Also,the sequence number in the returned program can be specified with address “P”. In this case, search starts from the relevant program beginning and returns to the initially appeared program number.

Though other interruptions cannot be performed in the execution of user macro program interruption program. M99 may be used to clear this state. When M99 command is performed individually ,it is performed before the end of the execution of the foregoing programs. Therefore user program interruption is also active for the last command of interruption program. If it is not applicable, user program interruption may be controlled with M96/M97.

The single block of M99, which only includes address O, N, P, L and M, the block is considered as the same block as the previous one of the program. Thus even a single block does not stop.

G××　X××××
M99;
It is actually identical with G××　X××××　M99; .
(They vary in whether G code is executed or not before M99.)

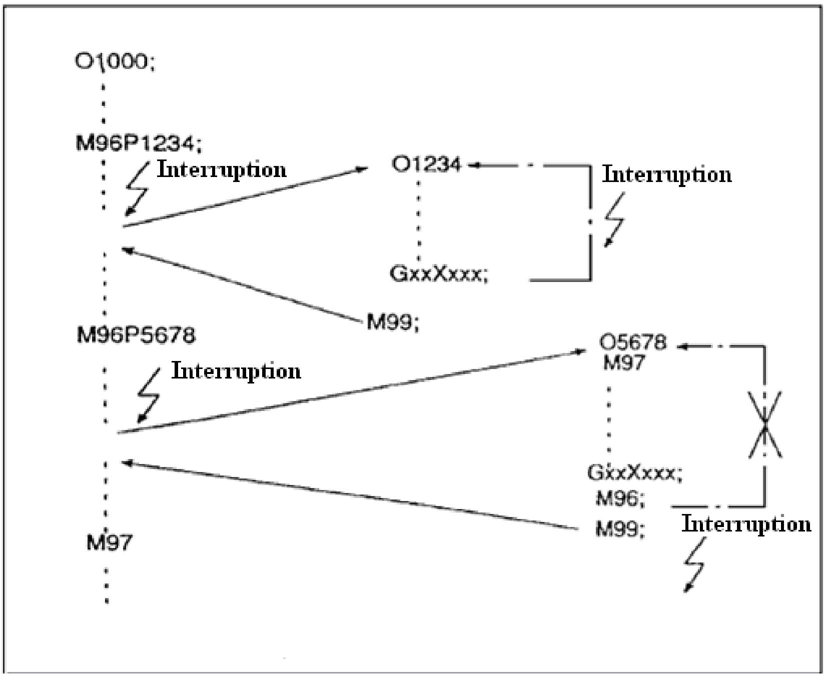


Fig. 3.6　Return from user macro program

Overlapping of user macro program interruption is not allowed during its execution. That is, other interruptions are automatically shielded in case of interruption. If M99 is executed, user macro program interruption will be active again. Here M99 is executed as a single block before the end of the previous block. In above example, the interruption in the G×× block of program O1234 is also active, and O1234 is executed again after interruption signal reinput. However, O5678 is under the control of M96/M97. Hence the interruption is active only after returning to O1000.

K11.3.7　User macro program interruption and modal information

User macro program interruption is different from general program calls. It is started by interruption signal (UINT) in program execution. As a rule, change of the modal information in an interruption program has a negative impact on the original program. Therefore, modal information restores the state before interruption when it returns to the original program with M99 even the modal information is changed in an interruption program.

When M99 P×××× is used to return to the original program from an interruption program, the modal information in a program is controllable. Hence the modal information changed in the interruption program may be contiuously used. (If the modal information at the interruption of the original program is reused, the return of the following travel depends on the modal data at interruption.)

Therefore, in this case:

1) Modal information is given in the interruption program.

or

2) Necessary modal information is commanded at return point.

Application is taken into account like this:

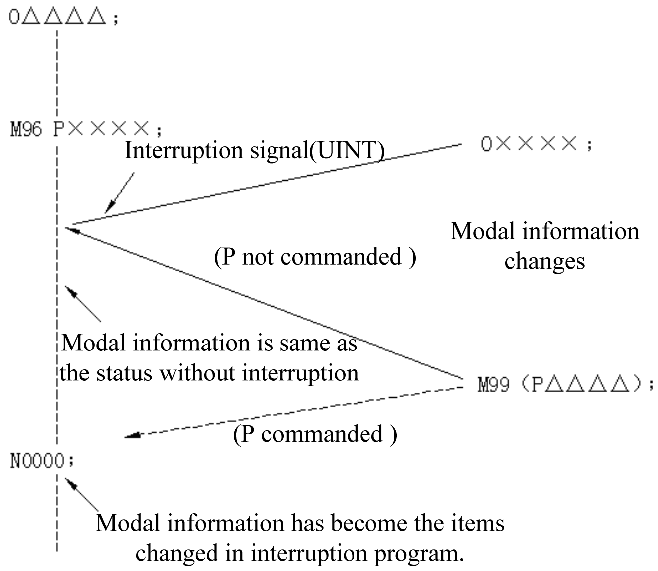


Fig. 3.7 User macro program interruption and modal information

While modal information is changed in interruption program:

1) During the return by M99

The modal information before interruption is active, while the modal information changed in interruption program is inactive.

2) During the return by M99 P△△△△,

The modal information changed in interruption program is also active after the returning (the same as M98).

K11.4 Parameters

					7	6	5	4	3	2	1	0
	0	2	5		MUSR		MSUB	MPRM				TSE

MUSR 1: Use macro program interruption function

0: Not use macro program interruption function

MSUB 1: Subprogram type user macro program interruption

0: Macro program type user macro program interruption

MPRM 1: The M code for the control of user macro program interruption is set by parameter.

0: User macro program interruption is under the control of M96 and M97.

Note: User macro program interruption is a part of User macro program B function. That is, user macro program interruption cannot be used without the user macro program B

selected.

The settings of parameters No.053 and 054 are active only when MPRM=1.

TSE 1: User macro program interruption is of state triggering mode.

0: User macro program interruption is of edge triggering mode.

Note: State triggering state is active in signal input ON state; edge triggering is active in the rise of signal.

	0	5	3	MACINTON

MACINTON: M codes for user macro program interruption active

Setting range: 03～97

	0	5	4	MACINTOF

MACINTOF: M codes for user macro program interruption inactive

Setting range: 03～97

Note: Parameters №053 and 054 are active only when MPRM (025 Bit4)=1,except this, they are inactive even though be set.

	3	1	4	MINT

MINT 1: The NC statement of interruption program starts to be executed until the end of block (user macro program interruption type II)

0: The NC statement of interruption program starts to be executed before the end of block (user macro program interruption type I)

Note: For type Ⅰ, once user macro program interruption signal (UINT) is input, the currently executing program is interrupted and the interruption program is executed. The motion after the return varies with the availability of NC statement in interruption program.

(Ⅰ) With NC statement

The remaining commands (amount of move and suspension time) disappear in the interrupted blocks.

(Ⅱ) Without NC statement

The remaining command in an interrupted block continues to execute.

However, the sending auxiliary function can be correctly output in both cases.

Type II: The current block is not interrupted and the interruption program is executed even user macro program interruption signal is input. If the interruption program has an NC statement, it starts to execute after the execution of the interrupted block.

K11.5 Diagnosis

					7	6	5	4	3	2	1	0
	1	2	0						UINT			

UINT 1: User macro program interruption ON.

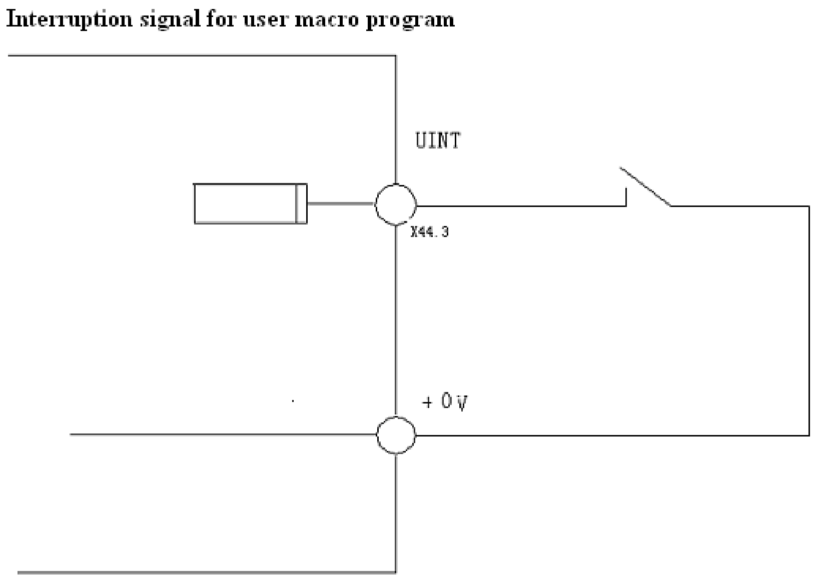
0: User macro program interruption OFF.

This signal is the external signal for user macro program interruption. Intended for the applications for high speed, the signal may be detected through not only the signal generated by strong power, but also external signal. Therefore, the manufacturer needs to connect the external signal to the position of the signal as shown below. The position is not applicable for other purposes.

					7	6	5	4	3	2	1	0
	0	4	4						UINT			

Note that the diagnosis numbers are #044 and #120.

K11.6 Internal Wiring Diagram



This function is optional, for implementing this function, X44.3 input in IO unit is needed

K11.7 Application Examples

K11.7.1 Processing for starting tool fault detection using external signal

<Specifications>

Malfunction restoration is immediately executed even in cycle movement. It performs only once in whole process.

<Parameter setting>

TSE=0: Edge triggering mode

MUSR=1: Enable user macro program interruption function

MSUB=*: Subprogram type/macro program type user macro program interruption

MPRM=*: Setting of the M codes for the control of user macro program interruption

MINT=0: Interruption program is executed before the end of block.

<Explanations>

User macro program interruption signal is ON during fault detection. It is kept on unless special operations are performed. Only one interruption is performed if the edge triggering mode is selected. Whether the interruption is executed or not is detected with diagnosis No.#120.

Parameters are set for the selection of subprogram type, macro program type and control M codes depending on programming.

K11.7.2 Inserting other job in continuous machining without interrupting the current program

<Specifications>

A short job is inserted in the program execution with longer machining time. It is troublesome that the original program restarts by general single block stop, MDI manual insertion.

< Parameter setting >

TSE=*: Selection of triggering mode for user macro program interruption

MUSR=1: Enable user macro program interruption function

MSUB=0: Macro program type user macro program interruption

MPRM=*: Setting of M codes for the control of user macro program interruption

MINT=1: Interruption program is executed after the end of block.

< Explanations >

Parameters are set as required for the triggering mode and the selection of M codes for user macro program interruption.

For user macro program interruption, interruption is prohibited in the execution of block and macro program type interruption is used to prevent the affection of the command segments in machining. The modal information, mechanical position, etc. in case of interruption in an interruption program are restored during the return to the original program so that the original program can be executed. If the interruption program is fixed, command M96 P×××× may be directly used. The interruption program is called by M98 P#100 if it is not fixed.

K11.7.3 Reading machining information in fixed intervals

< Specifications >

To manage machining status, machining information is sent out on a regular basis. This exerts no impact on machining sequence.

< Parameter setting >

TSE=0: Edge triggering mode

MUSR=1: Enable user macro program interruption function

MSUB=0: Macro program type user macro program interruption

MPRM=*: Setting of the M codes for the control of user macro program interruption

MINT=0: Interruption program is executed before the end of block

< Explanations >

Assuming that an interruption program does not include any NC statement, the user macro program interruption is repeated when the interruption signal is ON in state triggering mode; while the interruption program is started only once in edge triggering mode on a regular basis depending on the ON/OFF of interruption signal. Since block may interrupt in the midway, corresponding to the interruption signal rise it immediately interrupts.

The external output of machining information adopts user macro program DO to output modal information and position information.

An interruption program may execute in parallel with the original block. However, the machining will stop for a while at the end of the original block and before the end of the interruption program.

K11.7.4 Using the same program for general cutting and special cutting

< Specifications >

Each executive block is provided with special move. But this command is not used in general program.

< Parameter setting >

TSE=1: State triggering mode

MUSR=1: Enable user macro program interruption function

MSUB=*: Subprogram/macro program type user macro program interruption

MPRM=*: Setting of the M codes for the control of user macro program interruption

MINT=1: Interruption program is executed after the end of block.

< Explanations >

The interruption program shall be commanded as follows:

```
O××××;
M97;     ←———    Disable interruption
  :
M96;     ←———    Enable interruption
M99;
```

Therefore, user macro program interruption is executed at the end of each block in the original program. The special action to be performed is commanded beforehand in an interruption program. The program part that needs no user macro program interruption is disabled using M97.

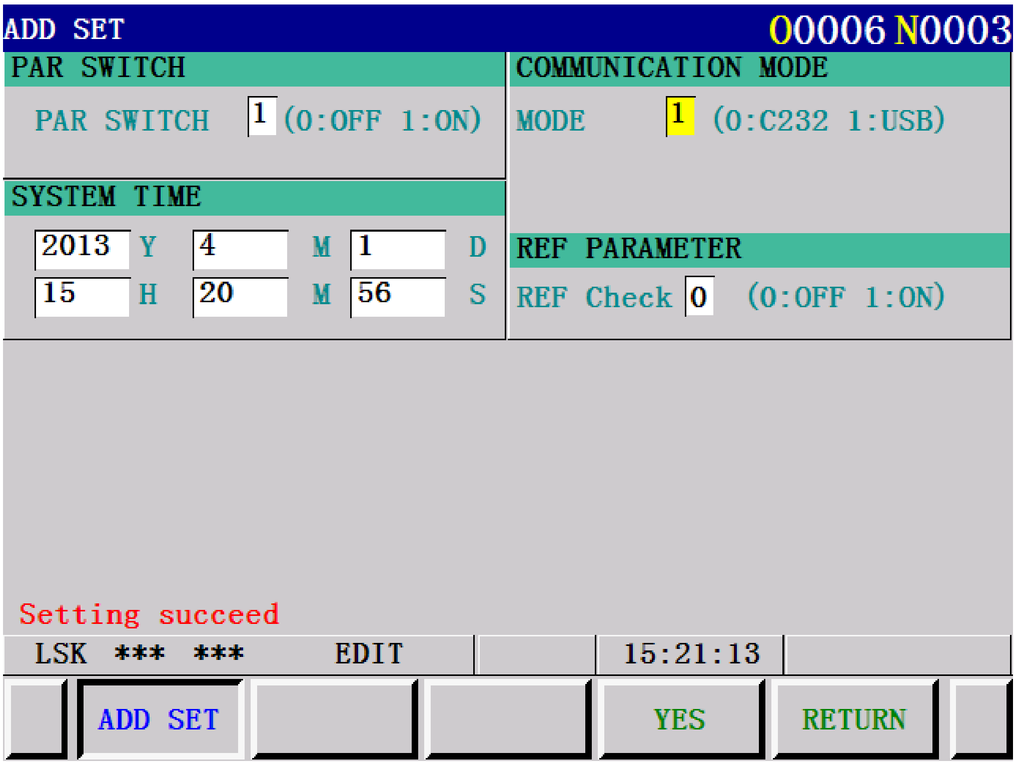
Appendix 12　USB interface transmission

L12.1　General

Parameters of NC and PLC, and process programs can be transmitted by USB interface; all the files transmitted are saved in folder"GSK983"under the root of U disk; when a file is transmitted from host unit to U disk, and finding no folder"GSK983"under the disk root, the system will add one then save the files into it.

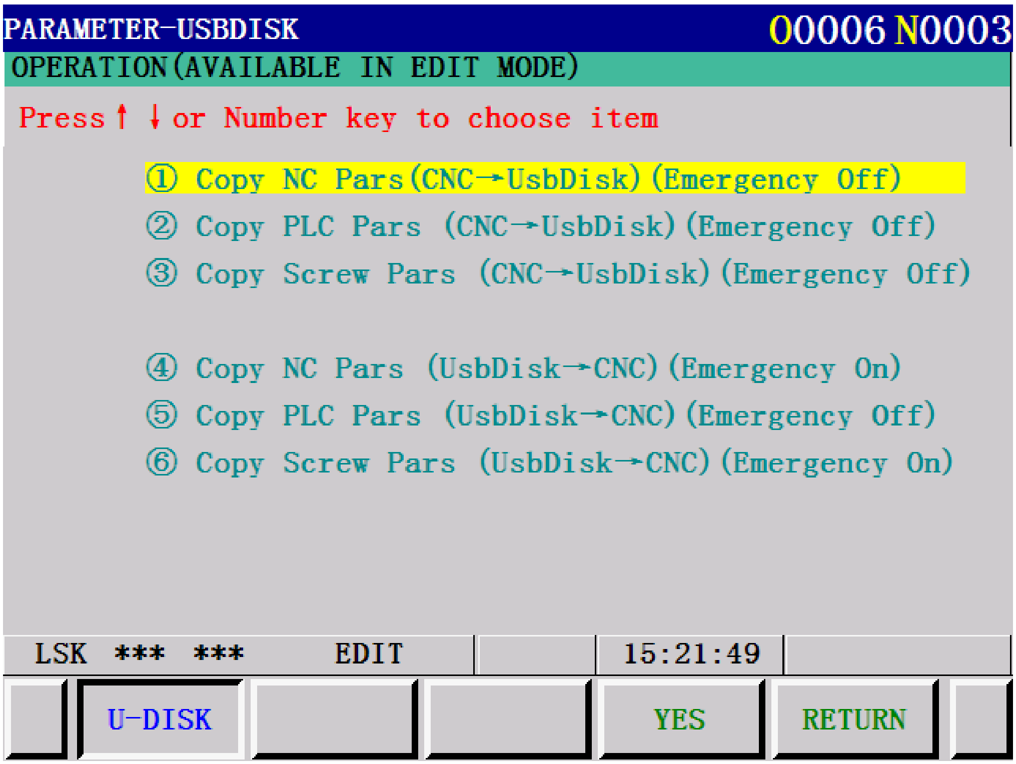
L12.2　Parameter setting of USB interface communication

As PARAMETER key is pressed,extended key + appears above the little keypad and by pressing it, the NC host unit displays U-DISK and ADD SET ;when add set key is pressed, the screen displays add set interface. Change the communication mode to 1(USB mode), press soft function key YES , and"setting succeed"is displayed. Setting communication mode is finished as follows:



L12.3　Parameters transmission operations by USB interfaces

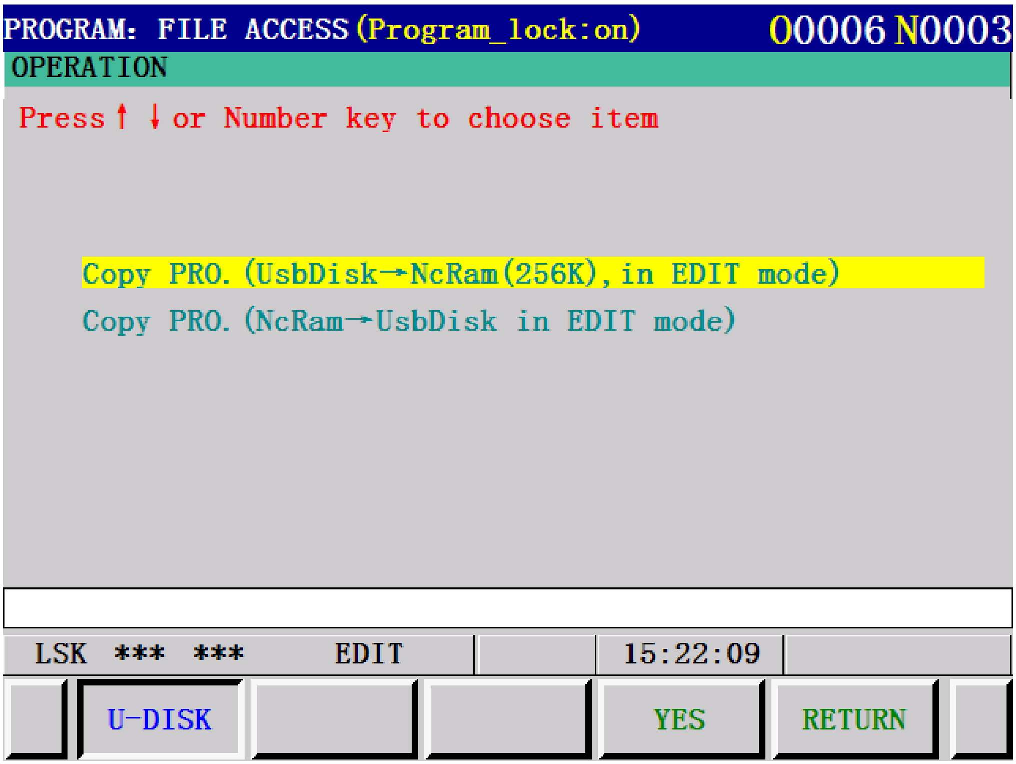
As PARAMETER key is pressed,extended key appears above the little keypad and by pressing it, the NC host unit displays U-DISK and ADD SET; when U disk is pressed, the screen shows the following operation options:



Users can select the operation options by moving the yellow bar as need and press YES to continue the corresponding operations; pressing RETURN key can return to previous menu.

L12.4 Workpiece programs transmission by USB interfaces

As PROGRAM key is pressed,小 extended key appears above the little keypad and by pressing it, the NC host unit displays U-DISK. Press it the screen shows the following operation options:



Users can select the operation options by moving the yellow bar as need and press YES to continue the corresponding operations;pressing RETURN key can reture to previous menu.

Note:

Only in the edit mode is the transmission enable!

Appendix13 C232 Serial Port Transmission

M13.1 General

GSK983Ta system communicates with PC by C232 serial port to transmit parameters, pitch compensations, offsets and workpiece programs. The operations are listed below:

Classification	Function	Mode	Operating procedures
Communication input	Parameter input	Emergency stop ON	P → - 9999 – data input
	Program storage	Edit	O → prg no. →data input
	Offset input	Edit	P→ -9999 →data input
	All program storage	Edit	O → - 9999 – data input
	parameter for pitch error compensation	Emergency stop ON	P → - 9999 – data input
Communication output	Parameter output	Edit	P → - 9999 - data output
	Offset output	Edit	P → - 9999 - data output
	Parameter for pitch error	Edit	P → - 9999 - data output
	All program output	Edit	O → - 9999 - data output
	One program output	Edit	O → prg no. – data output

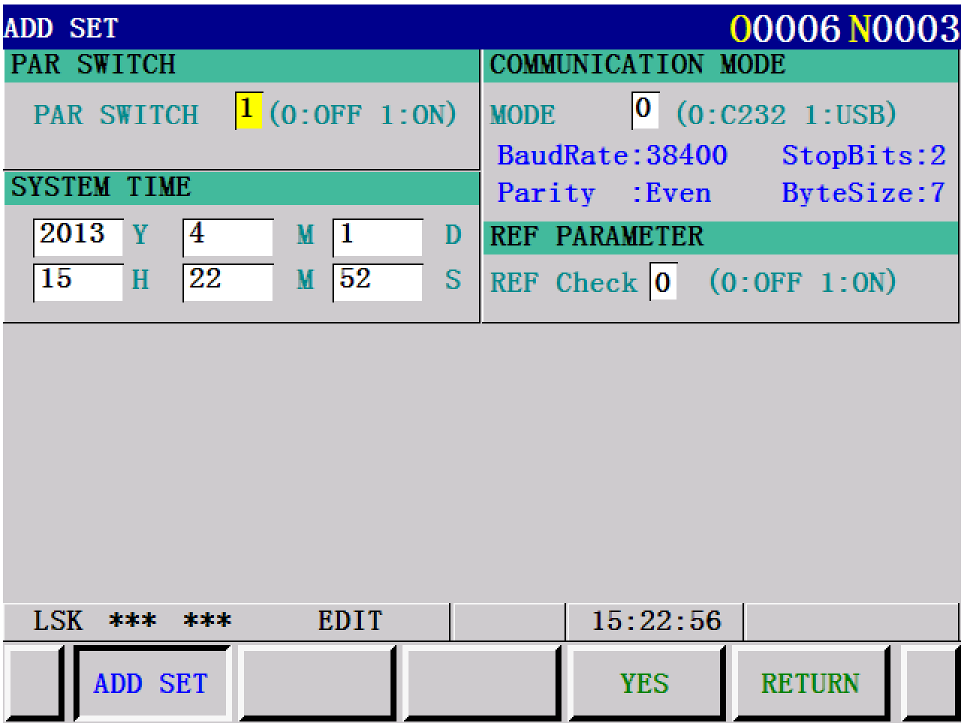
M13.2 Program transmission in C232 serial port communication mode

（1） Connection

Connect the GSK983Ta special communication cables to 9-pin serial port of PC and C232 interface in the 983Ta CNC operation panel.

（2） System setting

As PARAMETER key is pressed,extended key + appears above the little keypad and by pressing it, the NC host unit displays U-DISK and ADD SET; when add set key is pressed,the screen displays auxiliary setting interface. Change the communication mode to 0(C232 mode). Press soft function key YES, and“setting succeed”is displayed. Setting communication mode is finished as follows:



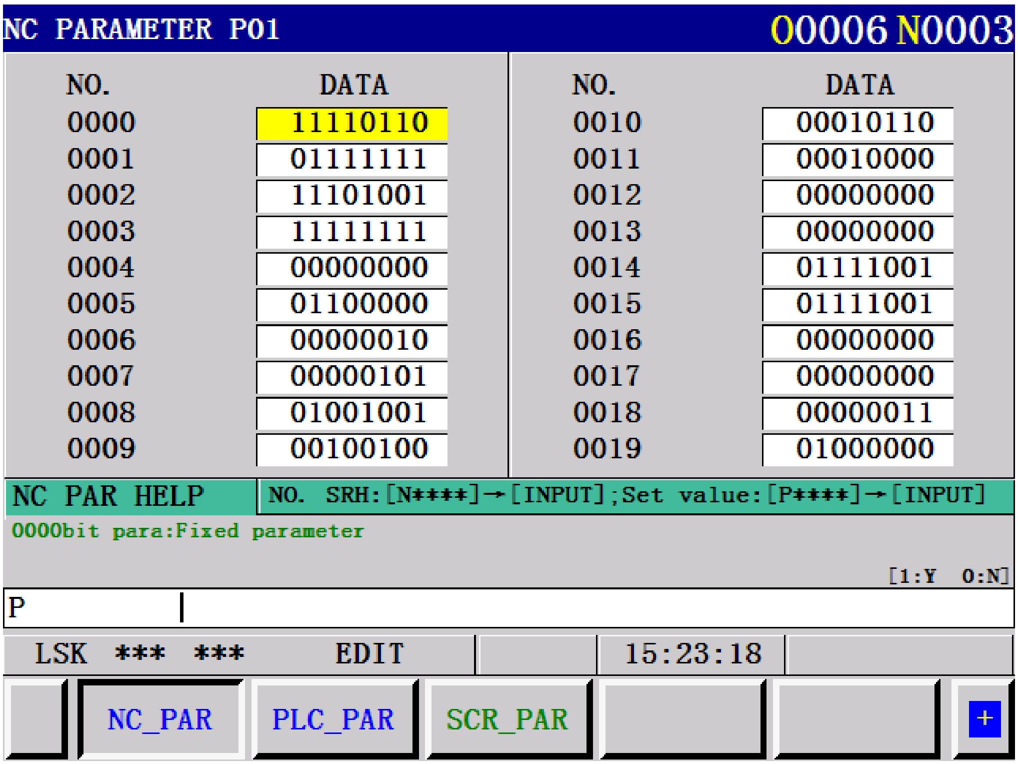
（3） As for PC, it can communicate data with CNC by RS232 interface after setting the communication software correspondingly.

M13.3　Parameter transmission in C232 serial port communication mode

（1） Set the communication mode of the CNC well according to the step (1)(2)in M13.2 chapter and so does the communication software setting of PC.

（2） Press parameter key and the CNC switches to parameter interface. In edit mode, input P-9999, and press data output key in the panel, at the moment, “EDIT” characters at the right of LCD screen bottom　twinkles and will disappear until parameters are sent.

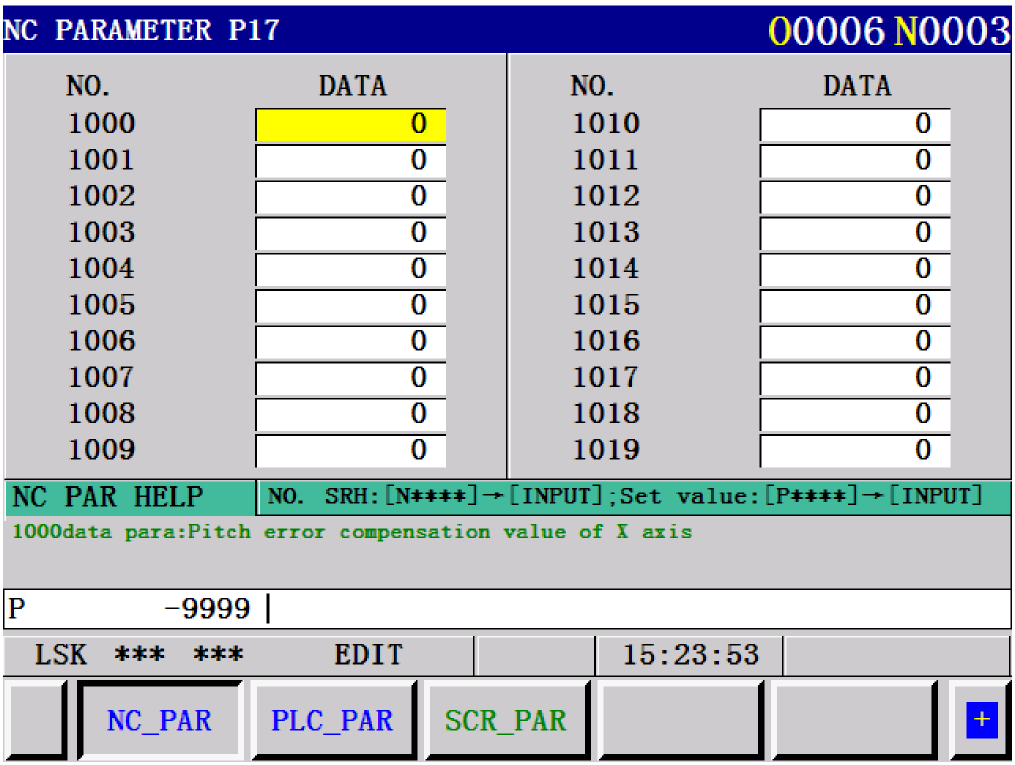
（3） To transmit parameter, press the emergency stop button then switch to parameter interface, input P-9999, press data input key in panel, choose parameter file by communication software of PC and select “send”. It is how parameters transmit to CNC.



Note: new parameter is active after restart.

（4） Transmit pitch compensation

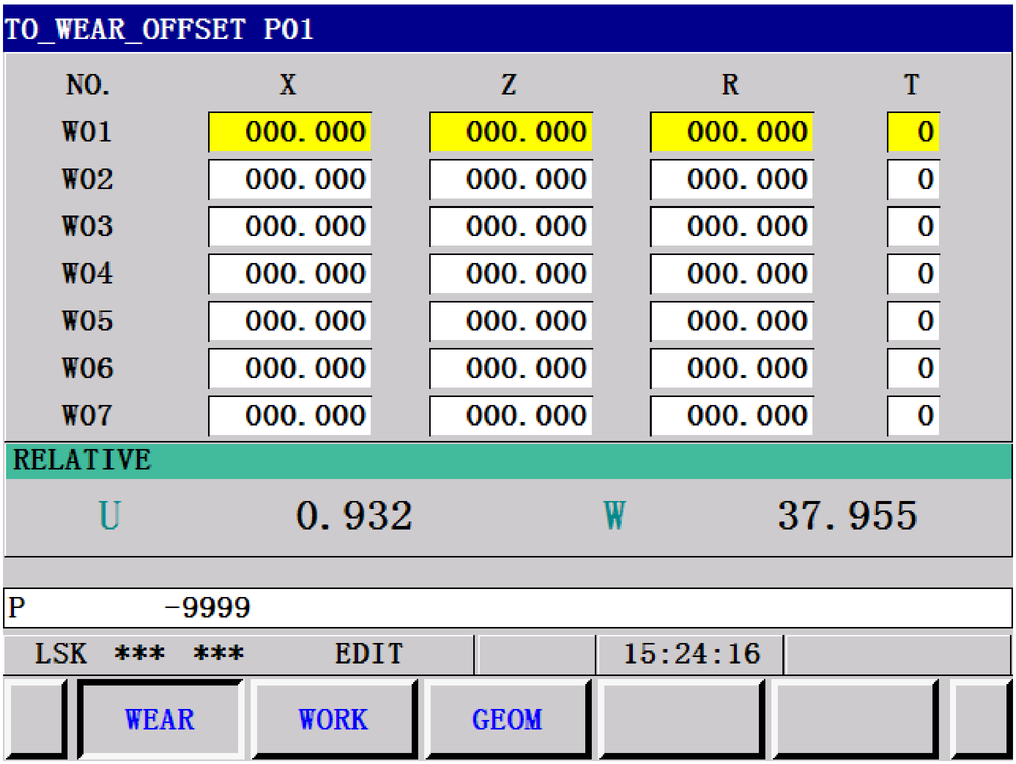
Pitch compensation is valid only when it is operated in NC1000# parameter interface. The operation procedure is like the one of parameter transmission as following:



Note: pitch compensation transmission should be performed in emergency stop mode, and it is active after restart.

（5） Transmit offset

Offset transmission should be performed in offset interface as follow shows, and the operation procedure is like the one of parameter transmission.



Appendix14 System clock setting and PLC programming

N14.1 System clock setting

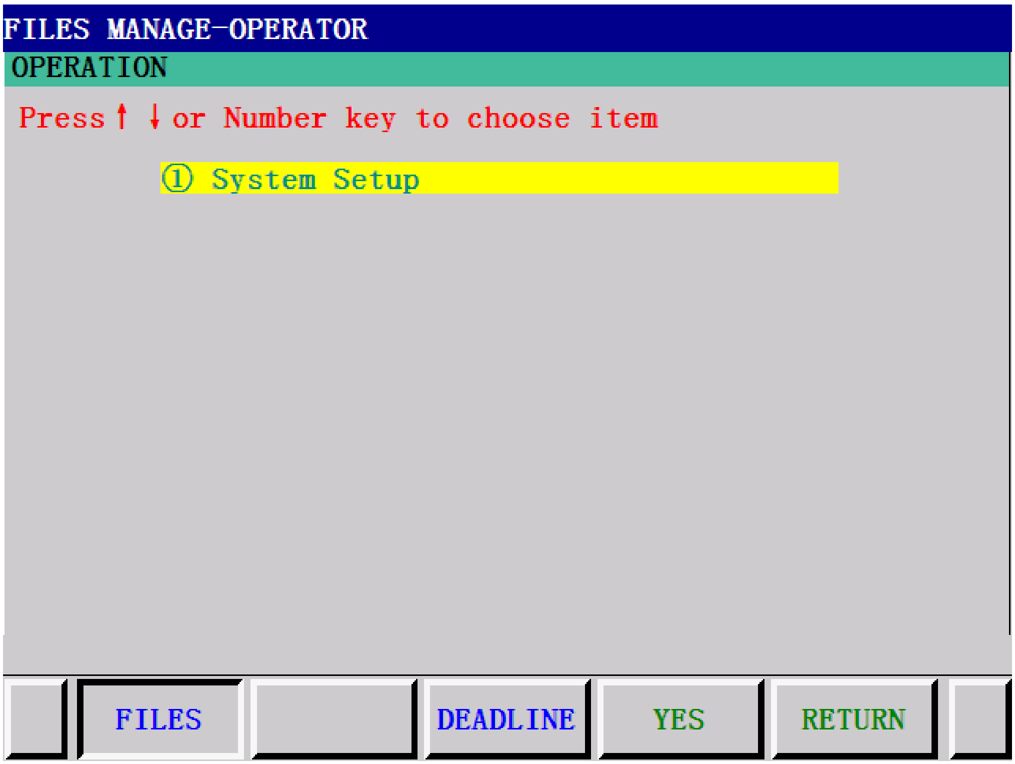
N14.1.1 Enter system clock management interface

Clock setting is in the system management interface that appears when the power is just on. When the system enters the version interface, press 

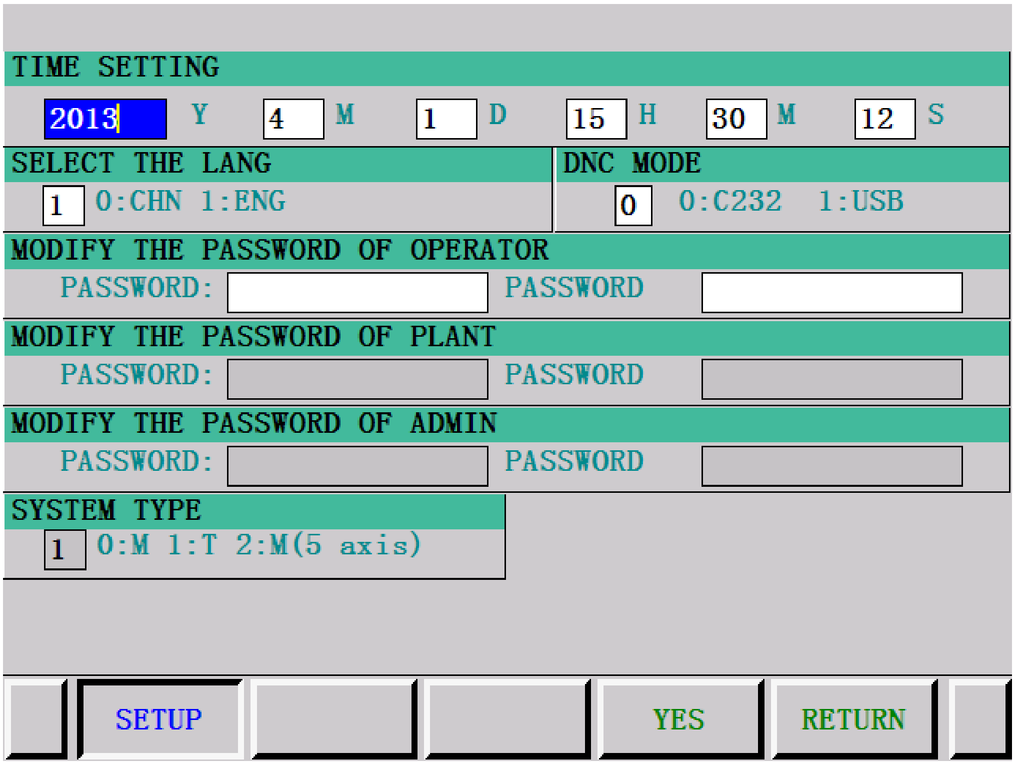
key and enter into the system management interface.

N14.1.2 Enter system setting interface

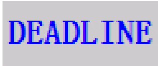
Input correct password of the level 1 user and press yes to enter the operation options for level 1 users.



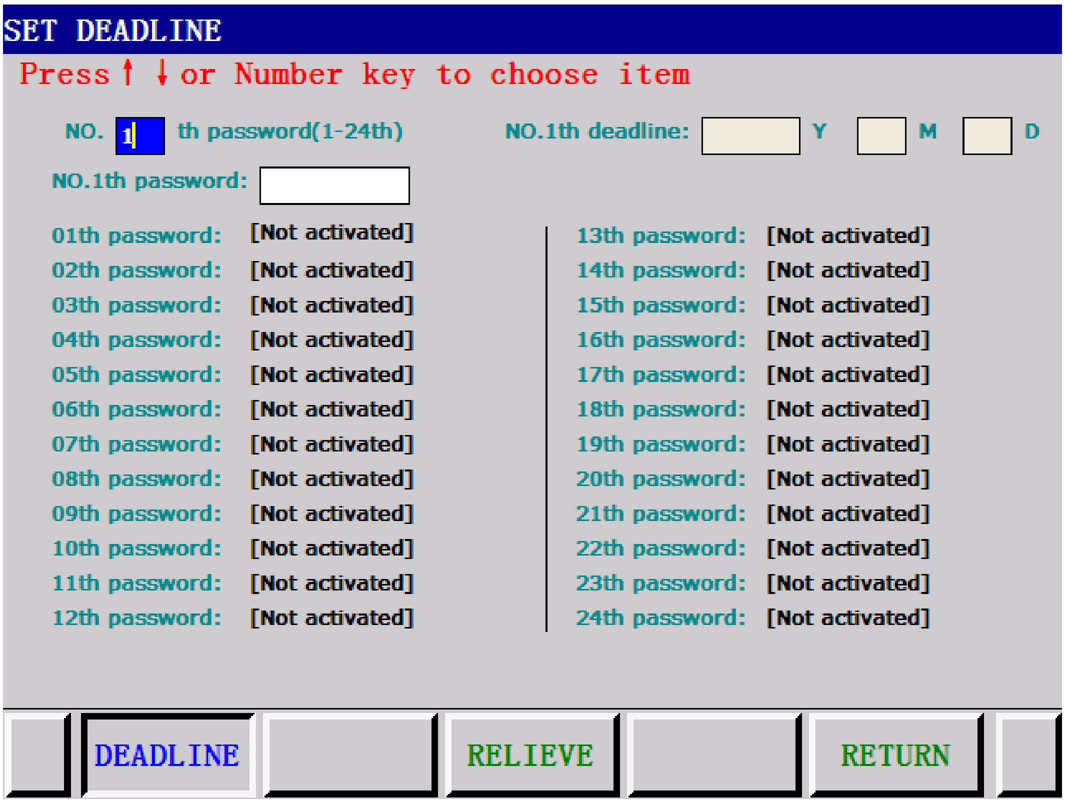
Press yes again to enter into system setting; different alter authority is given by the system automatically according to the user levels;after the alteration is done, press the set key to save the corresponding date into the system.



N14.1.3 Period function setting

Press soft function key 

and enter into period setting where user can disable the period password, as follows:

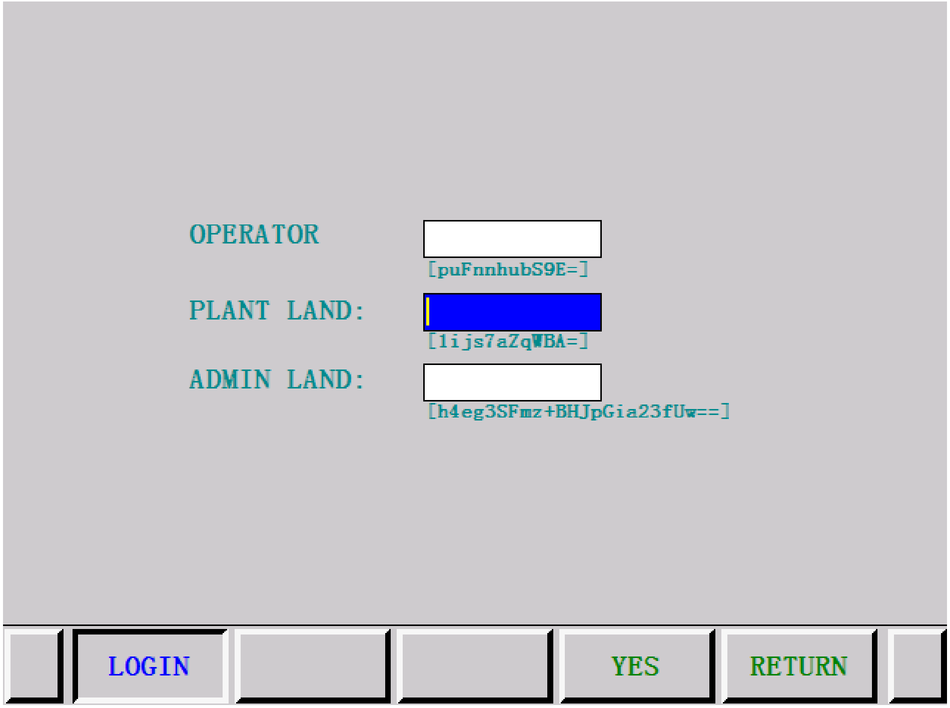


Note: the passwords of the period should be no more than 24.

N14.2 Level 2 user login interface

N14.2.1 Enter level 2 user login interface

At first press the shift key when the system is power on , enter into login, select level 2 user and input the right password as follows:

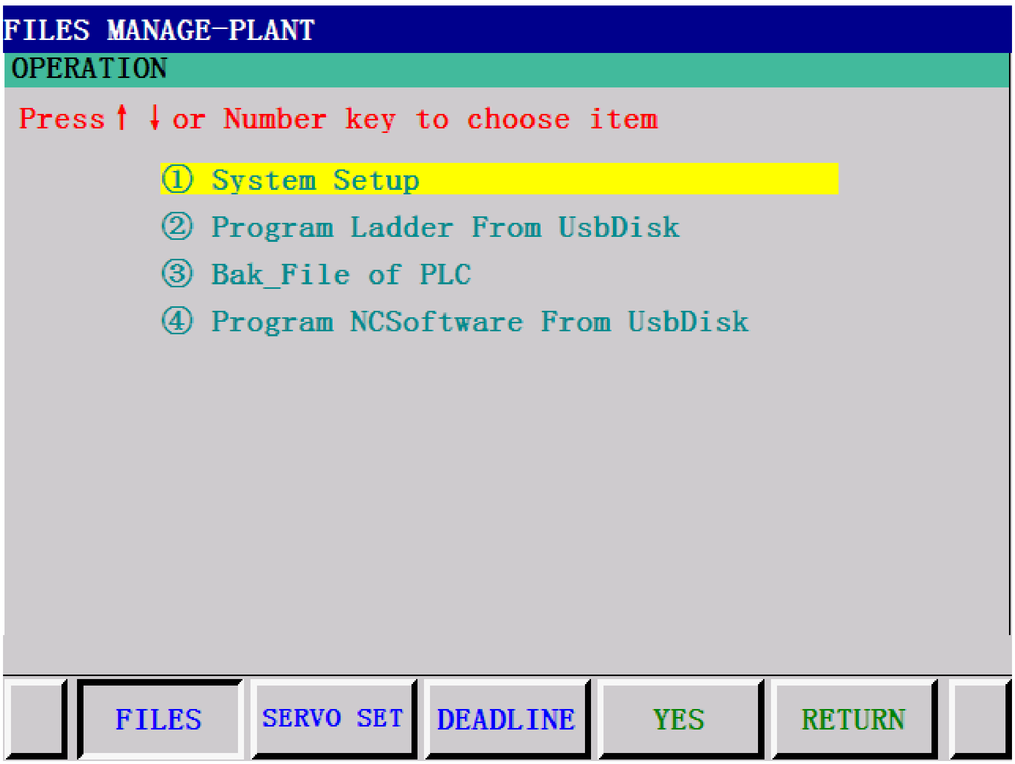


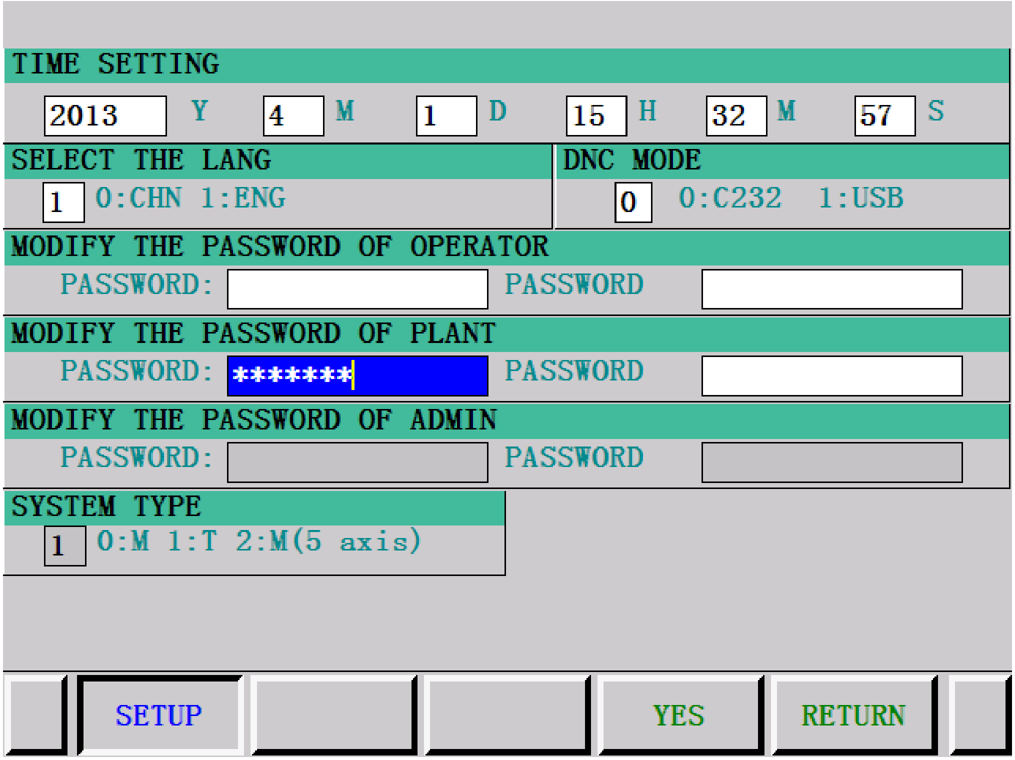
When the password is correct, system enters into operation options interface for the lever 2 users.

N14.2.2 PLC operation options interface

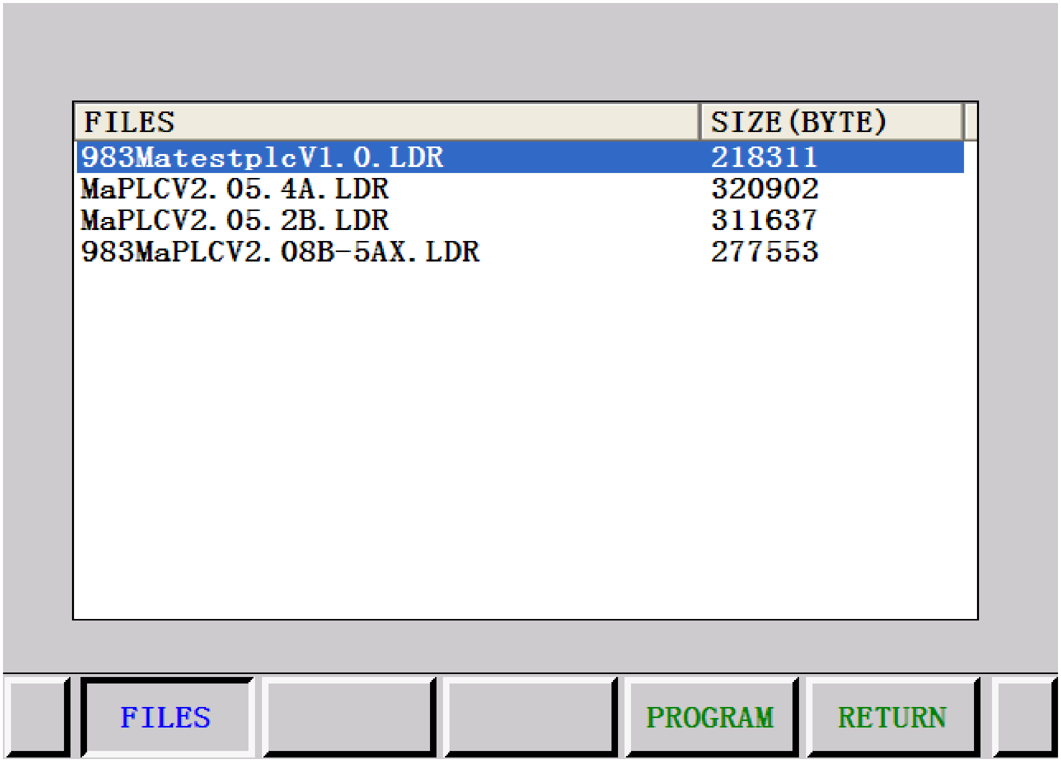
Move the yellow bar up and down to select corresponding options, or the level 2 authority , which includes system setting, PLC programming from U disk, and PLC files of CNCs;

(1)Contents system setting displays and operable by level 2 users;

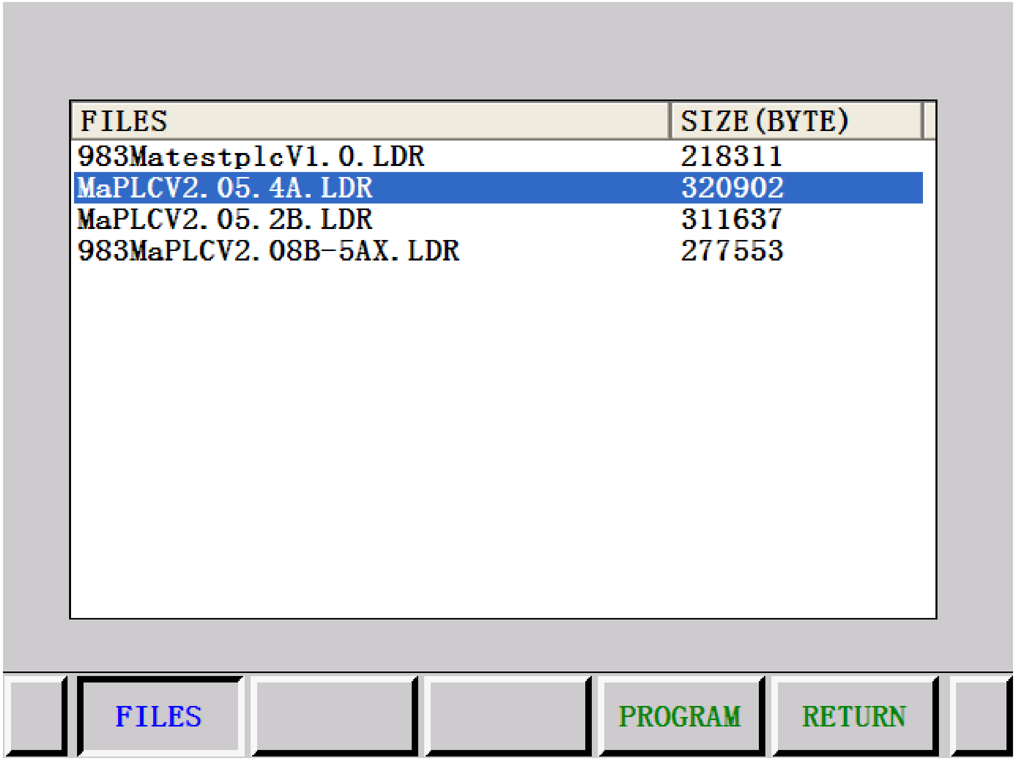




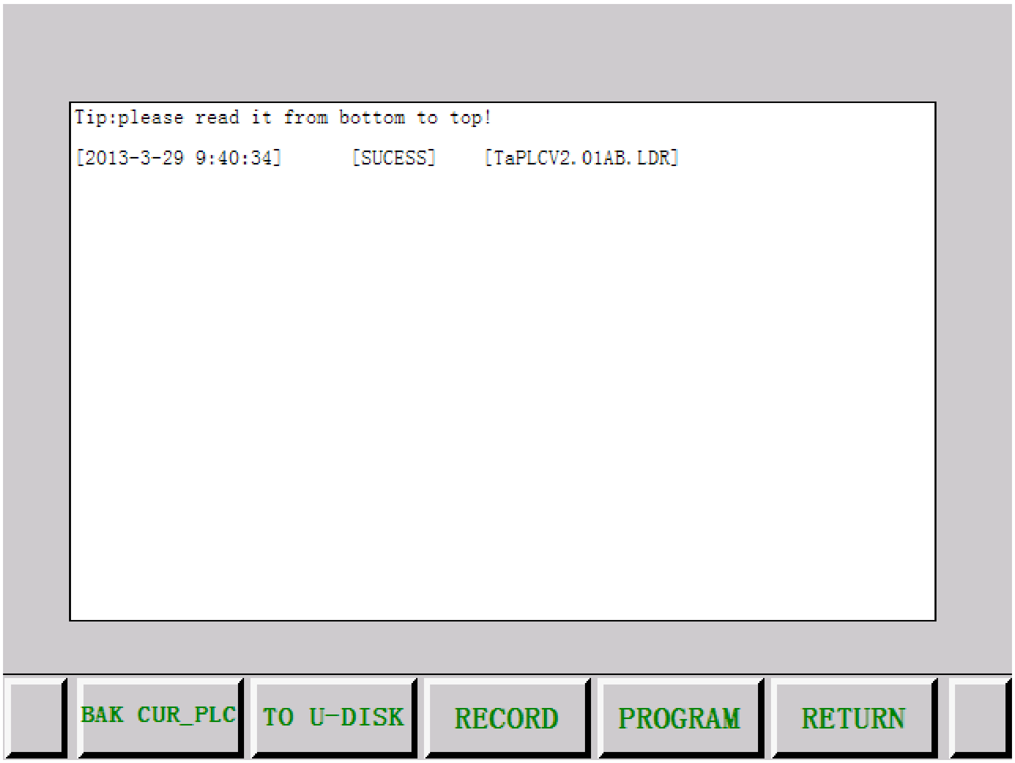
(2)Contents PLC programming from U disk displays and operable by level 2 users;



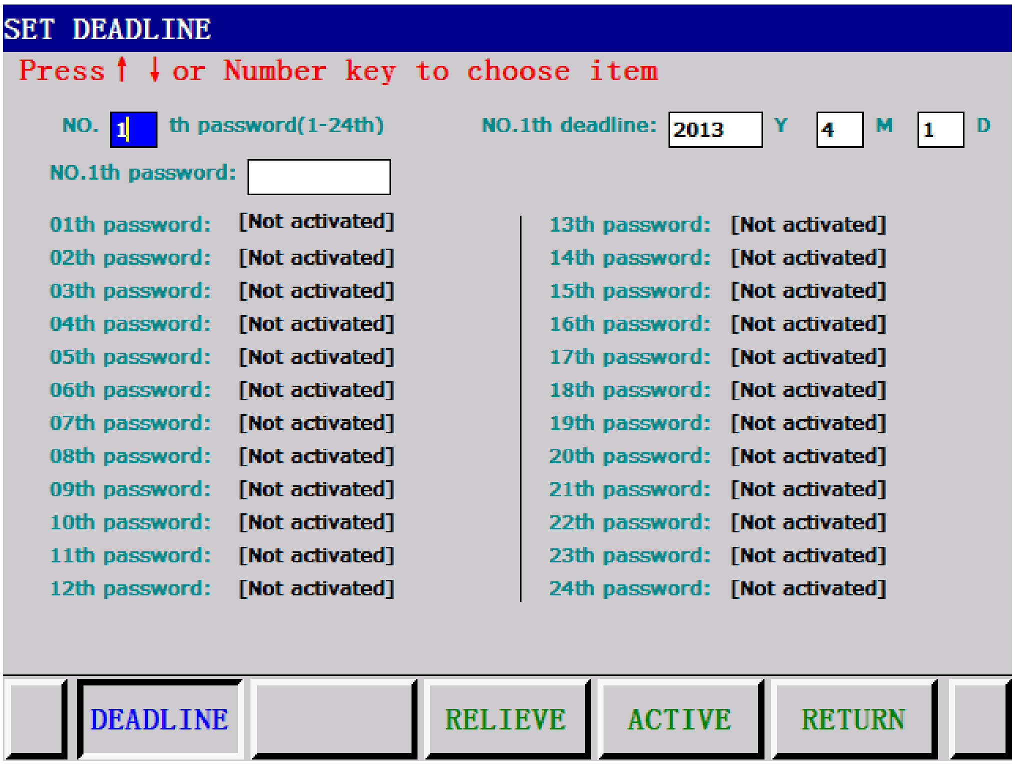
(3)Operations can be performed by level 2 users in PLC files of CNC;



(4)Programming log in level 2 authority;



(5)Press soft function DEADLINE key and enter period setting. The passwords of periods can be enabled and disabled by level 2 authority as follows:



(6) Press soft function key SERVO SET, and enter into servo and setting interface. The shielding of servo ready alarm and servo cut off alarm, and parameter switch can be set by level 2 authority as follows: