PROGRAMMING MANUAL

FOR

MAZAK

MAZATROL CAM T-2

EIA/ISO

66079

INTRODUCTION

The MAZATROL CAM T-2/T-3 is a conversation-type numerical control with a milling function. It is capable of using both the MAZATROL language originally developed by YAMAZAKI MACHINERY WORKS and world standard language EIA/ISO.

CAM T-3 can perform C-axis milling, drilling and the X- and Z-axis machining as an ordinary lathe.

This Programming Manual deals with the EIA/ISO program functions for lathes with MAZATROL CAM T-2.

This Manual is subject to change without notice for improvements of MAZATROL CAM T-2.

CONTENTS

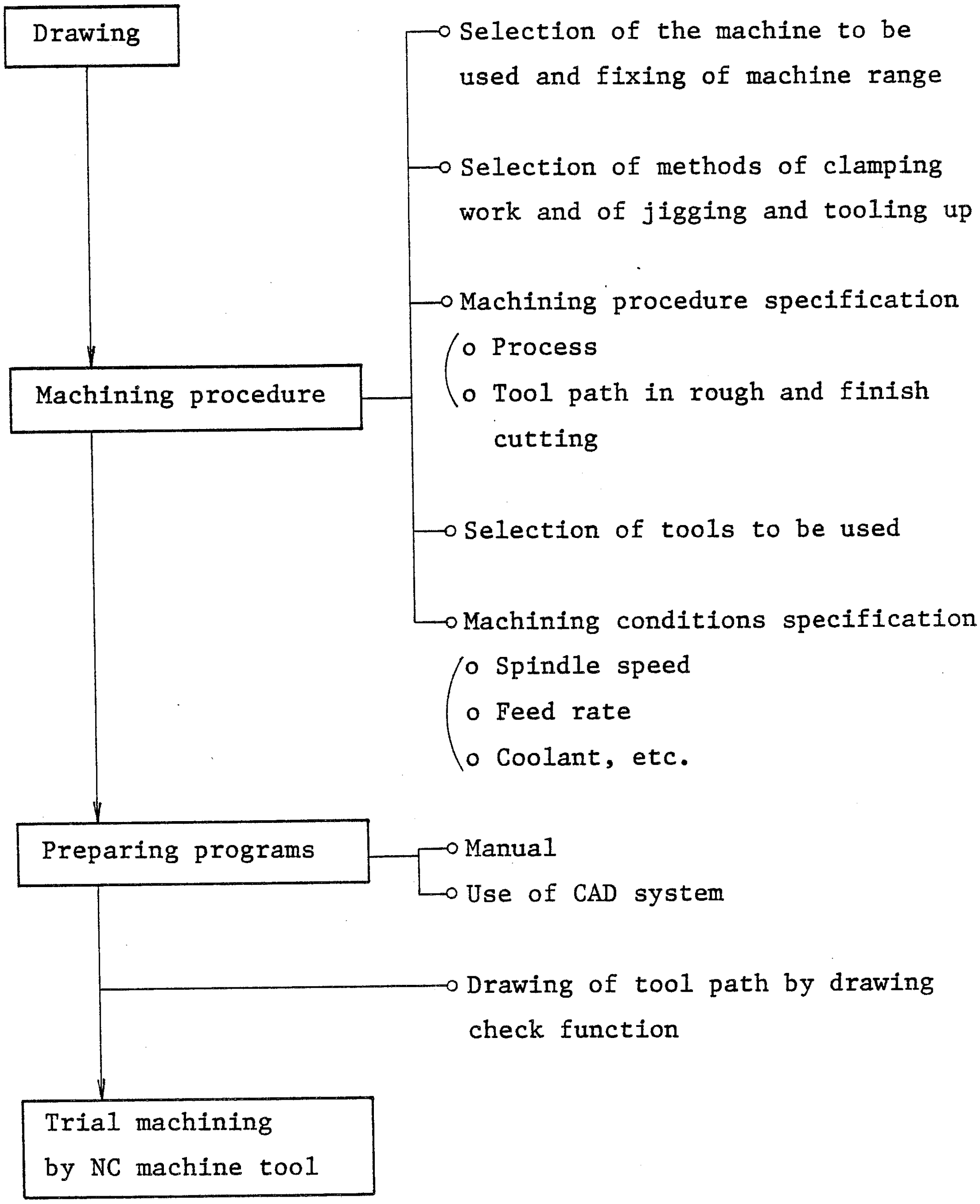
EIA/ISO PROGRAMMING

This Programming Manual deals with the EIA/ISO programs for general purpose MAZATROL CAM T-2.
Programming manuals for special models are available separately.

1. PROGRAMMING FOR MACHINING

Machining programs consist of a group of instructions for operating NC machine tools i.e. for transferring tools to desired positions and changing spindle speeds.

The following is the sequence of machine programming.



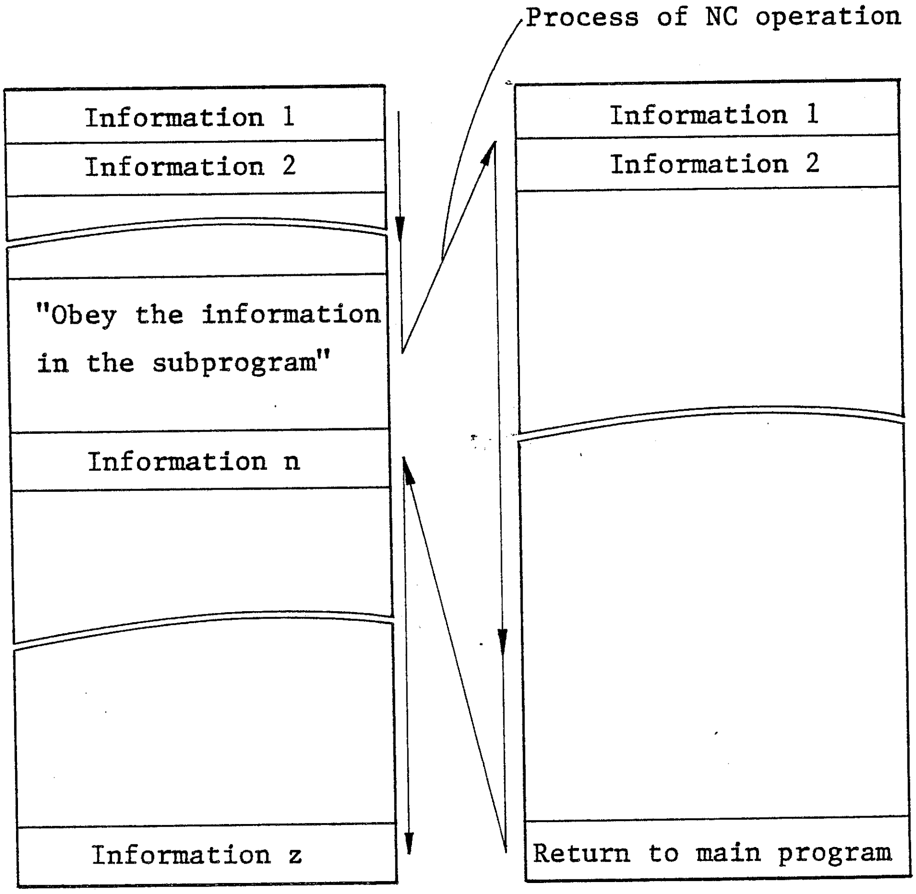
Machining programs are divided in EIA and ISO formats and will be referred to as EIA/ISO programs.

EIA/ISO programs are recorded on paper and cassette tapes and on floppy discs as well as in NC memory.

2. COMPOSITION OF EIA/ISO PROGRAM

The program is devided into the main program and the subprogram. Normally, the NC operates according to the information contained in the main program but when the command meaning "Obey the information in the subprogram" is encountered on the main program, the NC obeys the information in the subprogram thereafter. When the command meaning "Obey the information in the main program" is encountered in the subprogram, the NC obeys the information in the main program thereafter.

Thus, NC begins by running the main program and even when using subprograms halfway, NC returns to the main program to complete the total machining operation.



Main program Subprogram

Note) Refer to Chapter 10. SUBPROGRAM for details.

2.1 Block

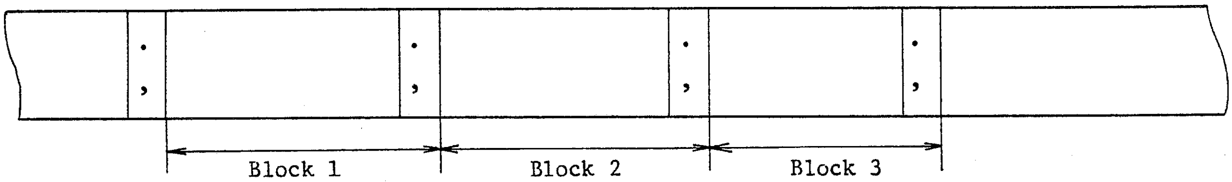
The program is composed of several commands.

One command is called a block.

One block is discriminated from another block by an end of block code.

This manual expresses the end of block code by the symbol ; in the following.

Machining program:

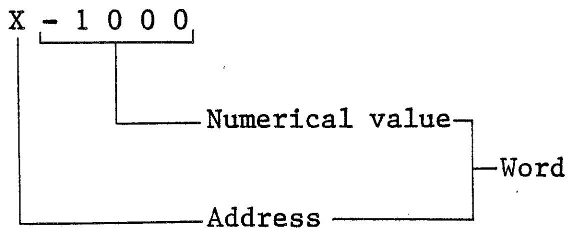


Note 1) The maximum number of characters per block is 64 including $\boxed{(}$, $\boxed{,}$ and $\boxed{)}$.

Note 2) The end of block code is "CR" in EIA code and "LF" in ISO code.

2.2 Word

A block is composed of one or more words. A word is composed of an address followed by numbers as shown below. (Algebrical sign (+ or -) may be added before a numerical value.)



The address is a letter which prescribes the meaning of the numerical value following the address.

The addresses and their meanings are as follows.

Some addresses may vary their meanings depending on the preparatory functions (G code) specified in the program.

Note) Refer to Chapter 7. for preparatory functions (G code).

Table of address

Function	Address	Meaning
Program number	:(ISO)/O(EIA)	Program number
Sequence number	N	Sequence number
Preparatory function	G	Motion mode (Linear, arc, etc.)
Coordinate word	X, Z U, W	Motion command of coordinate axes
	R	Arc radius, corner R
	I, K	Coordinate values of arc center, chamfering amount
Feed function	F, E	Feedrate, thread lead
Spindle function	S	Spindle speed
Tool function	T	Tool number, tool offset number
Miscellaneous function	M	ON/OFF control on the machine tool
Dwell	P, U, X	Dwell time
Program number designation	P	Designation of the subprogram number
Sequence number designation	P, Q	Designation of the sequence number at the program repetitive location
Repetitive count	L	Repetitive count in subprogram and drilling cycle
Parameter	A	Angle of canned cycle
	D, I, K, Q	Cutting depth of canned cycle
	D	Number of canned cycles

2.3 Input Format

Each word in a block must be commanded with the fixed format as follows. The input format is called variable block format in which the number of the word in a block and the number of characters in a word are permitted to be changed. This format is very convenient for programming

(1) Input in metric

$$N04.G02.\alpha L{+}043.\beta L{+}043.\gamma L{+}043.\left\{\begin{matrix}RD043\\ ID043.K043\end{matrix}\right\}.\left\{\begin{matrix}F032\\ F050\\ E034\end{matrix}\right\}.S04.T4.M02;$$

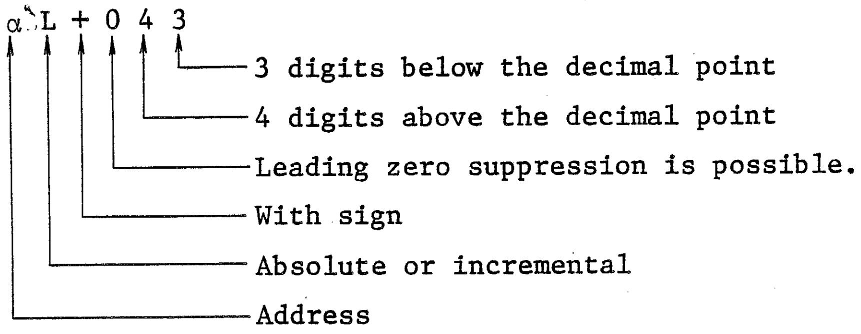
(2) Input in inch

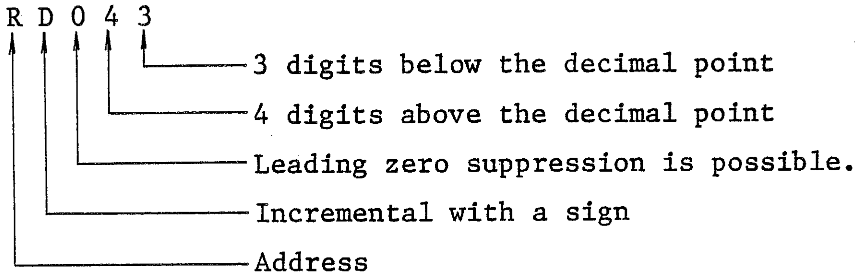
$$N04.G02.\alpha L{+}044.\beta L{+}044.\gamma L{+}053.\left\{\begin{matrix}RD034\\ ID034.KD034\end{matrix}\right\}.\left\{\begin{matrix}F024\\ F032\\ E016\end{matrix}\right\}.\ S04.$$

$$.\left\{T4\right\}.M03;$$

Note 1) α,β: X or U, : Z or W and

Note 2) Above addresses and numerical values are meant as follows





Note 3) F032 (millimeter input) and F024 (inch input) are input formats for feed per revolution.
F050 (millimeter input) and F032 (inch input) are input formats for feed per minute.

Note 4) The above format omits address A, P, Q, L and D, because they have various meanings.

Note 5) Refer to item 2.4 for decimal point programming.

Note 6) Selection of millimeter or inch input depends on $\boxed{\text{P6}}$ setting. Programs are not changeable using G20/G02.

2.4 Decimal Point Programing

Numerical values with a decimal point can be inputted in programming. A decimal point may be used with mm, inch, second or speed values. However, some addresses cannot use a decimal point. Decimal points are placed to indicate metric, inch or second decimal position.

Example:

Z15.0 ..... Z15 mm or Z15 inch.
F10.0 ..... 10 mm/rev, 10 mm/min, 10 inch/rev or 10 inch/min.
G04X1. .... Dwell for one second

The addresses with which a decimal point can be used are as follows:

X, Z, U, W, I, K, R, F and E

Note 1) In dwell command, decimal point programming for addresses X and U is possible but it is not possible with address P.
(This is because address P is also used for sequence number.)

Note 2) If a decimal point position changes with the G code, the appropriate G code should be commanded before the numerical values are specified in one block.

Example 1: G98; (mm/min designation)

F1.0 G99; ...... 0.01 mm/rev is assumed.
(G99 is mm/rev designation)

G98; (mm/min designation)

G99 F1.; ...... 1 mm/rev is assumed.
(G99 is mm/rev designation)

Note 3) Values with and without a decimal point can be commanded together.

X1000 Z23.7; and X1.0 Z33700; mean the same.

Note 4) Values less than the least input increment are deleted if specified.

2.5 Maximum Programmable Values

The maximum programmable values of each address are listed in table below. Note that these figures give the maximum numerical limit, not the mechanical limit in the NC machine tool. When editing programs, machine limitations must be understood before fixing command values. Cutting feed rates which are the same should be checked against the specifications when preparing the programs.

Table Basic Addresses and Commandable Range

Function	Address	Meaning	Commandable range	Remarks
Program number	: (ISO) O (EIA)	Program number	1-9999	
Sequence number	N	Sequence number	1-9999	
Preparatory function	G	Designation of motion mode	0-99	
Coordinate word	X, Z	Motion command of coordinate axes (absolute)	±9999.999mm ±999.9999inch	X in diameter
	U, W	Motion command of coordinate axes (incremental)		U in diameter
	R	Arc radius, corner R		
	I, K	Coordinate values of arc center, chamfering amount		
Feed function	F	Feed per minute	1-15000mm/min 0.01-600.00inch/min	
	F	Feed per revolution, thread lead	0.01-500.00mm/rev 0.0001-50.0000inch/rev	
Thread lead	E	Precision thread lead	0.0001-500.0000mm 0.000001-9.999999inch	
Spindle function	S	Main spindle speed	0-9999	
Tool function	T	Tool number, tool offset number	0-6464	Tool offset amount ±999.999mm ±99.9999inch
Miscellaneous function	M	ON/OFF control on the machine tool	0-999	
Dwell	X, U P	Dwell time	0-9999.999 0-9999	
Program number designation	P	Designation of subprogram number	1-9999	
Sequence number designation	P, Q	Designation of sequence number at the program repetitive location	1-9999	Q is also used for "Parts count available/not available".

Function	Address	Meaning	Commandable range	Remarks
Repetitive count	L	Repetitive count in subprogram	1-9999	
Parameter	A	Angle of canned cycle	0-127	
	D,I,K,Q	Cutting depth of canned cycle	±9999.999mm ±999.9999inch	For D and Q values, decimal point programming is not possible.
	D	Number of canned cycles	1-9999	
Note 1) Tool function	A	Return tool number	0-64	
	B	Next tool number	0-64	

Note 1) This function is available for special models.

2.6 Program Number

This control can store programs in NC memory. The program number is used to differentiate one program from another.

The program number is identified as follows:

Program number $O\underbrace{\square\square\square\square}$

4 digits (1 - 9999, zero cannot be used.
Leading zero suppression is possible.)

A program begins with a program number and finishes with ER(EIA), %(ISO), M02;, M30; or M99;.

01234 .... M02;	05678 .... M30;

Program No. 1234 (main program)   Program No. 5678 (main program)

M02; and M03; above mean the end of main program, and M99; means the end of a subprogram.

09012 .... M99;	

Subprogram No.9012

Note 1) In ISO code, colon ":" can be used instead of "O".

Note 2) A block with an block skip code such as /M02;, /M30; or /M99; is not regarded as the end of a program.

2.7 Sequence Number

A sequence number can be specified with up to 4-digit number (1-9999) following address N at the head of a block. The order of sequence numbers is arbitrary and need not be consecutive. Also sequence numbers can be specified in all blocks or in the blocks in which it is required.

It is recommended that sequence numbers be specified sequencially and be specified at important points such as at the block in which a tool is changed and a new tool is used.

2.8 Block Skip

With a slash followed by a number (/n (n=1-9)) programmed at the beginning of a block and with the BLOCK SKIP n menu (n=1-9) set ON, information of the block with /n of a number corresponding to BLOCK SKIP n menu is ignored.

With the BLOCK SKIP n menu set OFF, information of the block with /n is valid.

The following range is ignored with the BLOCK SKIP 3 menu ON and the next block operation is performed.

```
; /3N1234 G00 X4 ..........; N1235 ......;
        Range ignored
```

Note 1) "/1" can be omitted as "/".

Note 2) If a slash (/) code is placed elsewhere in the block, the information from the slash (/) code to the ";" code is ignored.

Note 3) When storing the program into NC memory, this function is ineffective. The block with slash (/) code is also stored in the memory irrespective of ON/OFF of BLOCK SKIP menu key.

Note 4) When punching out the program from the memory, the program is punched out irrespective of ON/OFF of BLOCK SKIP menu key.

Note 5) The block skip is identified when the information is read into the buffer storage from memory. When a block preceded by slash has been read into the buffer, it is ignored even if the BLOCK SKIP menu key is turned off.

BLOCK SKIP:

Search the EIA/ISO program by the PROGRAM SEARCH function and display the POSITION, COMMAND and TRACE pictures.
When the M (Menu selection) key is pressed, the next menu is produced.

BLOCK SKIP 1	BLOCK SKIP 2	BLOCK SKIP 3	BLOCK SKIP 4	BLOCK SKIP 5	BLOCK SKIP 6	BLOCK SKIP 7	BLOCK SKIP 8	BLOCK SKIP 9
		↑			↑			

/3N ...; and /6N ...; blocks are skipped.

3. COORDINATE WORD

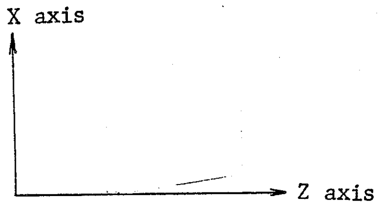
A coordinate word specifies a tool movement and is composed of the address of the axis to be moved and the value indicating the move direction and amount. The value varies depending on the absolute or incremental programming.

3.1 Controlled Axis

The axes which can be controlled by this NC among the movable axes on the machine are called controlled axes. Each controlled axis is specified by the address of the coordinate word used on this NC. The cotrolled axes of the machines to T-2 specification comprise X and Z axes, both of which can be controlled simultaneously.

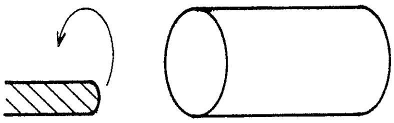
Coordinate axes are used as a reference in preparing machining programs. Note the following two points when preparing machining programs.

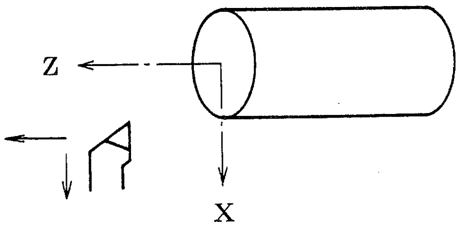
(1) Programs must refer to the standard coordinate system (the right-hand cartesian coordinate system).



" ⟶ " indicates positive direction.

(2) When programming, assume that the workpiece (hereinafter referred to as work) stands still and the tool is moved around the work.





3.2 Increment System

The increment system is determined by the following two factors:

(1) Least input increment (Input unit)

The minimum unit of tool travel inputted in program editing. This is given in mm or inch.

(2) Least command increment (Output unit)

The minimum unit of tool motion, given in mm or inch. Either one of the following combinations is used:

			Least input increment	Least command increment
Linear axis	mm input	Feed screw in metric	0.001 mm	0.001 mm
	inch input		0.0001 inch	0.001 mm
	mm input	Feed screw in inch	0.001 mm	0.0001 inch
	inch input		0.0001 inch	0.0001 inch

In diameter programming, the least command increment of the X-axis is also the diameter value.

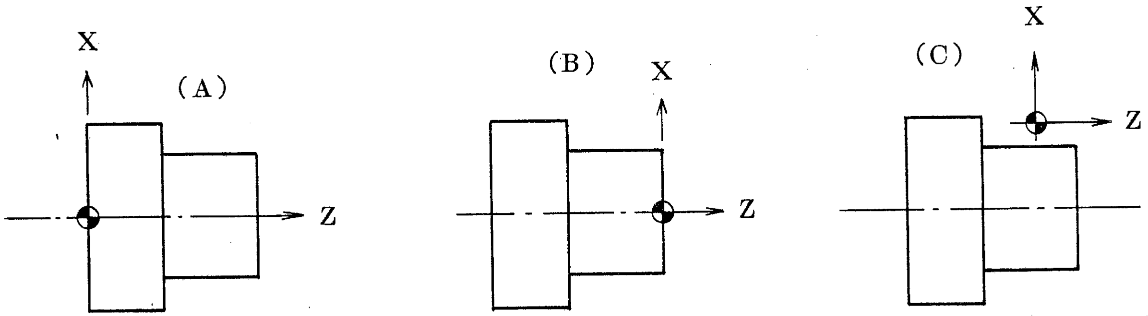
3.3 Program Zero Point, Coordinate System and Starting Point

When programming, a program zero point and a coordinate system must be determined. Program zero point can be placed at an arbitrary position.

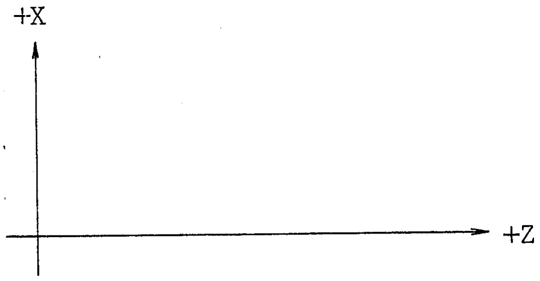
Usually, X axis should be coincided with the work center and Z axis should be coincided with the work's left or right face. See figures (A) and (B) below.

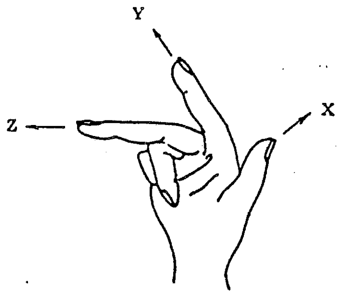
No coordinate system such as in (C) is adopted.

There is no program zero point for C axis (specify direction and turning degree when necessary).



This coordinate system is called work coordinate system.

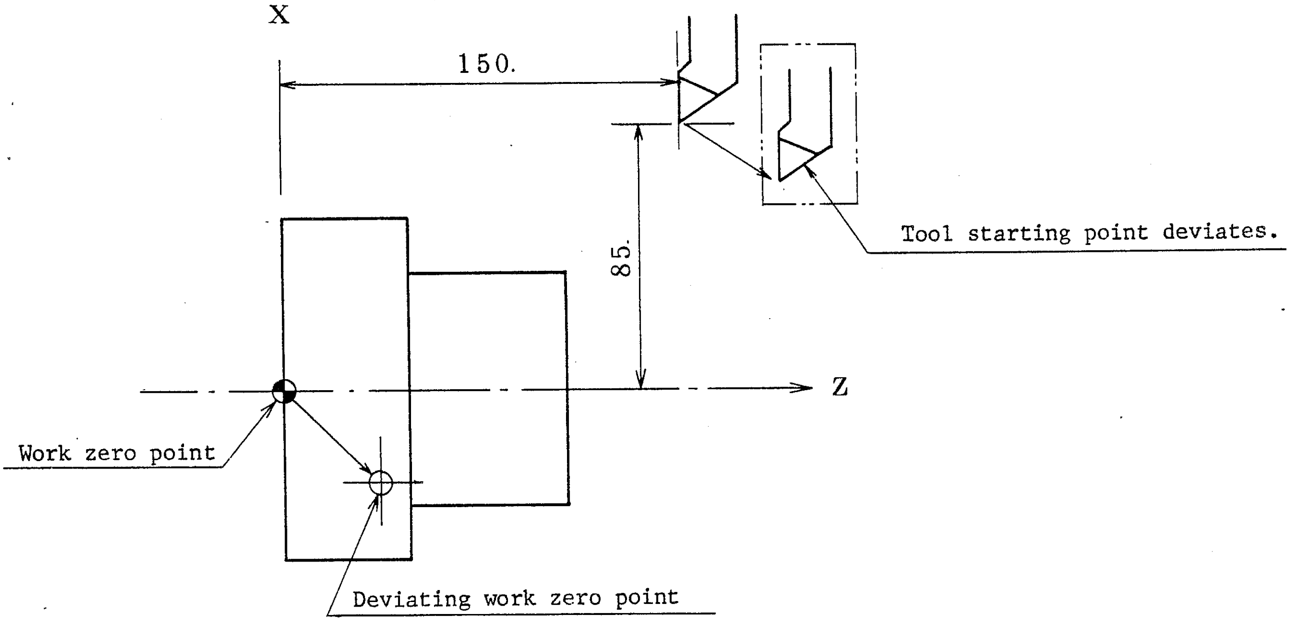




The right hand

Set the coordinate system when programming by using absolute instructions. Values handled are those of the established coordinate system. When setting up the coordinate system, generally specify the coordinate values of the tool starting point.

Example:



In the above case, the command

G50 X170.0 Z150.0;

sets up a coordinate system with reference to the center of spindle rotation and the left end of work set to X0 and Z0.

Note) X axis; diameter value instruction (refer to 3.6).

The deviation of the tool starting point causes a corresponding deviation in the set-up coordinate system.
Refer to 7.1 "Setting of Coordinate System" of this Manual for further details.

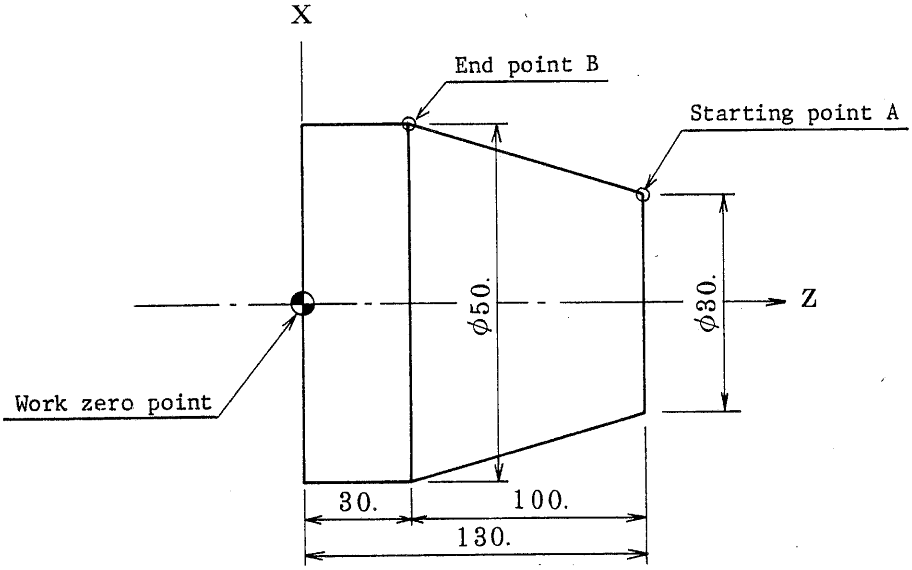
3.4 Zero Point

The zero point is a fixed position on a machine tool. The function of zero point return will return the tool to the zero point. Accordingly, a program may not start from a certain position in the work coordinate system but may start from the zero point. In this case, because the zero point is a certain point on the machine tool and the program is made according to the work coordinate system in which the zero point is on the work, the position of the tool returned to the zero point must be commanded by a G50 command in the work coordinate system.

If CYCLE START is commanded before completion of the X- or Z-axis return to the zero point, usually an alarm will result. But CYCLE START can be made valid using A5 parameter even before completion of the X- or Z-axis return to the zero point.

3.5 Absolute and Incremental Programming

Absolute or incremental programmings are given to specify the relative tool moving position with respect to the program zero point or to the tool moving distance respectively.



In above case, absolute and incremental programming relationships are tabulated below.

Command method		Address	Command of movement from A to B in the above
Absolute programming	Specifies an end point in the work coordinate system	X (Coordinate value of X axis Z (Coordinate value of Z axis	X50.0 Z30.0;
Incremental programming	Specifies a distance from starting point to end point	U (Distance along X axis) W (Distance along Z axis)	U20.0 W-100.0;

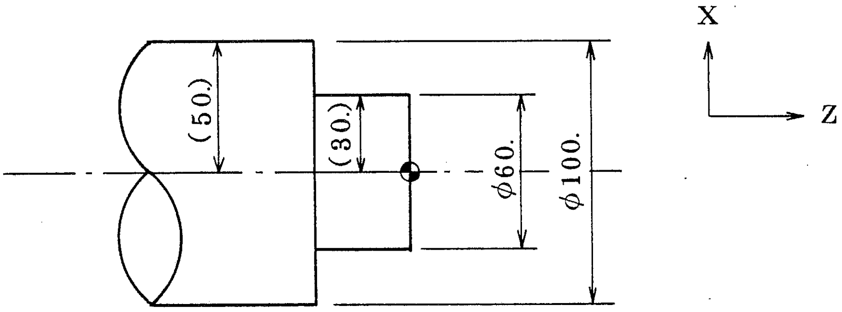
Note 1) Absolute and Incremental commands can be used together in a block. In the above example, a command as follows is possible:

```
X50.0 W-100.0;
U20.0 Z30.0;
```

Note 2) When both X and U or W and Z are used together in a block, the one specified later is effective.

3.6 Diameter Programming

Lathe work is generally circular except when milling. Therefore, use diametric values for the X-axis instruction.



Large diameter X100.0

Small diameter .... X 60.0

Use double values (diametric values) as coordinate values (50, 30).

Note) The coordinate system must be set up with the center of spindle rotation as X0.

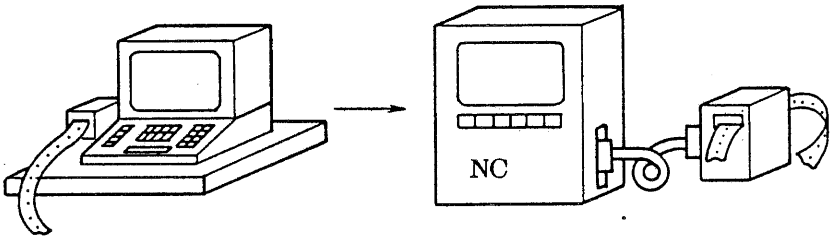
When using the diameter programming, note the conditions listed in the table.

Item	Notes
Z-axis command	Irrespective of diameter or radius.
X-axis command	Commands with diameter value.
Incremental command by address U	Commands with diameter value.
Coordinate system setting (G50)	Specifies a X-axis coordinate value with diameter.
X component of tool offset value	Commands diameter value.
Parameters in G90-G94, and G70-G76 such as cutting depth along X axis, (D,I,K)	Commands radius value.
Radius designation in circular interpolation (R,I,K)	Commands radius value.
Feedrate along X axis	Change of radius/rev Change of radius/min
Display of X-axis position	Displayed in diameter value.

4. INPUT OF EIA/ISO CODE PROGRAMS

The means for inputting, editing and ejecting the MAZATROL CAM T-2 EIA/ISO code programs can be divided in the following.

(1) Paper tape

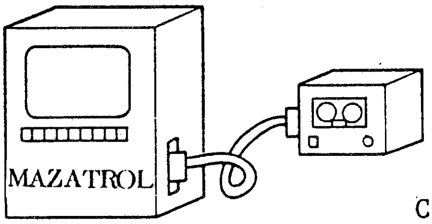


Paper tape punch (commercially available)

Paper tape input and output unit (special accessory)

o Transfer of information from paper tape to MAZATROL
o (MAZATROL provides free editing by keys.)
o Paper tape is punched to record MAZATROL information and is removed.

(2) Magnetic tape (cassette)



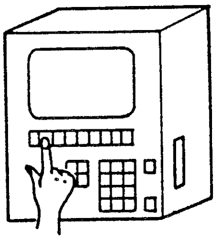
Cassette deck (special accessory)

o Transfer of information from cassette tape to MAZATROL
o (MAZATROL provides free editing by keys.)
o Cassette tape records the MAZATROL information and is removed.

← ↓ → ↑

↓
65
4A

(3) Direct input



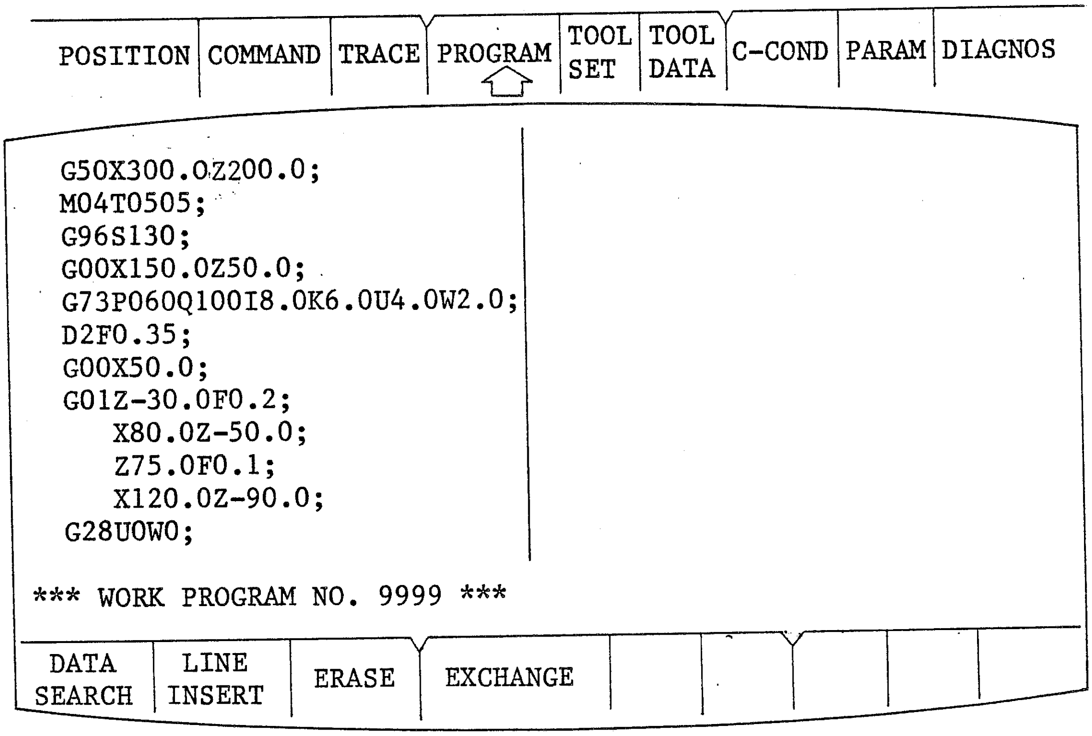
MAZATROL

- o Direct input from program manuscripts using menus and number keys
- o MAZATROL provides free editing by keys.
- o Input is possible up to the limit of the MAZATROL storage capacity.
- o Unused programs are erased.

4.1 EIA/ISO Programs

4.1.1 Call of displays

Press the key (display selection) for the display selection menu and press the PROGRAM menu key.



```
G50X300.0Z200.0;
M04T0505;
G96S130;
G00X150.0Z50.0;
G73P060Q100I8.0K6.0U4.0W2.0;
D2F0.35;
G00X50.0;
G01Z-30.0F0.2;
   X80.0Z-50.0;
   Z75.0F0.1;
   X120.0Z-90.0;
G28U0W0;
```

*** WORK PROGRAM NO. 9999 ***

EIA/ISO program display

4.1.2 Preparing and editing EIA/ISO programs

(i) Preparing

Search for new work numbers and press the PROGRAM menu key.

WORK NO.	PROGRAM		↑ ↑	↓ ↓	CHECK	SIMULATN	LAYOUT	PROGRAM FILE

Next press the displayed EIA/ISO PROGRAM menu key.

EIA/ISO program preparation mode will then be obtained.

	EIA/ISO PROGRAM	MAZATROL PROGRAM						

In the preparation mode, the following menus appear.

(Address menu)

G N 0 /	X U C Q	Z W H B	F E A J	S R D V	T I L Y	M K P (	EOB SP INS)	CHANGE

(1)	(2)
(3)	(4)

Use address menus and number keys to prepare programs. Then, press the CHANGE menu key. Addresses will then change in the (1) to (4) order. Press ADDRESS menu key. Return will then occur to (1). Press the ⇨ (INPUT) key at proper intervals within 64 characters for one block. The contents of the editor lines will then transfer to the program display (WORK PROGRAM NO.). Press ▭ key (menu selection). Program preparation mode will then be terminated and change to edit mode.

(ii) Edit

EIA/ISO program displays are automatically obtained when the work numbers of EIA/ISO programs are searched for.

WORK NO.	PROGRAM		↑ ↑	↓ ↓	CHECK	SIMULATN	LAYOUT	PROGRAM FILE
	Press							

Press the PROGRAM menu key. Edit mode will then be obtained.

4.1.3 Edit mode operation

(Edit mode menu)

DATA SEARCH	LINE INSERT	ERASE	EXCHANGE					

(Address menu)...Change over among 4 modes by the CHANGE menu key.

G N 0 /	X U C Q	Z W H B	F E A J	S R D V	T I L Y	M K P (	EOB SP INS)	CHANGE

(i) DATA SEARCH (the designated character strings after the cursor at the program list range are searched for.)

(1) Transfer the cursor to search start position and press DATA SEARCH menu key.

(2) Press the number keys and address menu key to enter the character strings to be searched for in the edit range.

(3) Press the ➔ (INPUT) key.

⟶ The cursor will transfer to the character string searched for.

(ii) INSERT (a desired character string is inserted at the position of the cursor at the list range.)

(1) Transfer the cursor to insertion place and press the LINE INSERT menu key.

(2) Press the number keys and address menu key to enter in the edit range the character strings to be inserted.

(3) Press the ➔ (INPUT) key.

⟶ The character strings at the edit range will be entered at the position of the cursor in the list range.

(iii) ERASE (the character strings at a desired range are erased from the position of the cursor at the list range.)

(1) Transfer the cursor to erasure start position and press the ERASE menu key.

(2) Transfer the cursor to erasure completing position.

(3) Press the ⇨ (INPUT) key.

→ The character strings between the cursors will be erased.

(iv) EXCHANGE (the character strings at a block where the cursor is positioned are exchanged with other characters.)

(1) Transfer the cursor to a block for exchange and press the ERASE menu key.

→ The block to be subjected to exchange is transferred to the edit range.

(2) Exchange the characters at the edit range using the number keys or address menu key.

(3) Press the ⇨ (INPUT) key.

→ The character strings at the list range will be exchanged with those at the edit range.

Note 1) No. "0" number is not displayed in EIA/ISO program display because it is registered as work number on the NC memory. It is therefore not necessary to give any work number to the head of programs using the address "0".

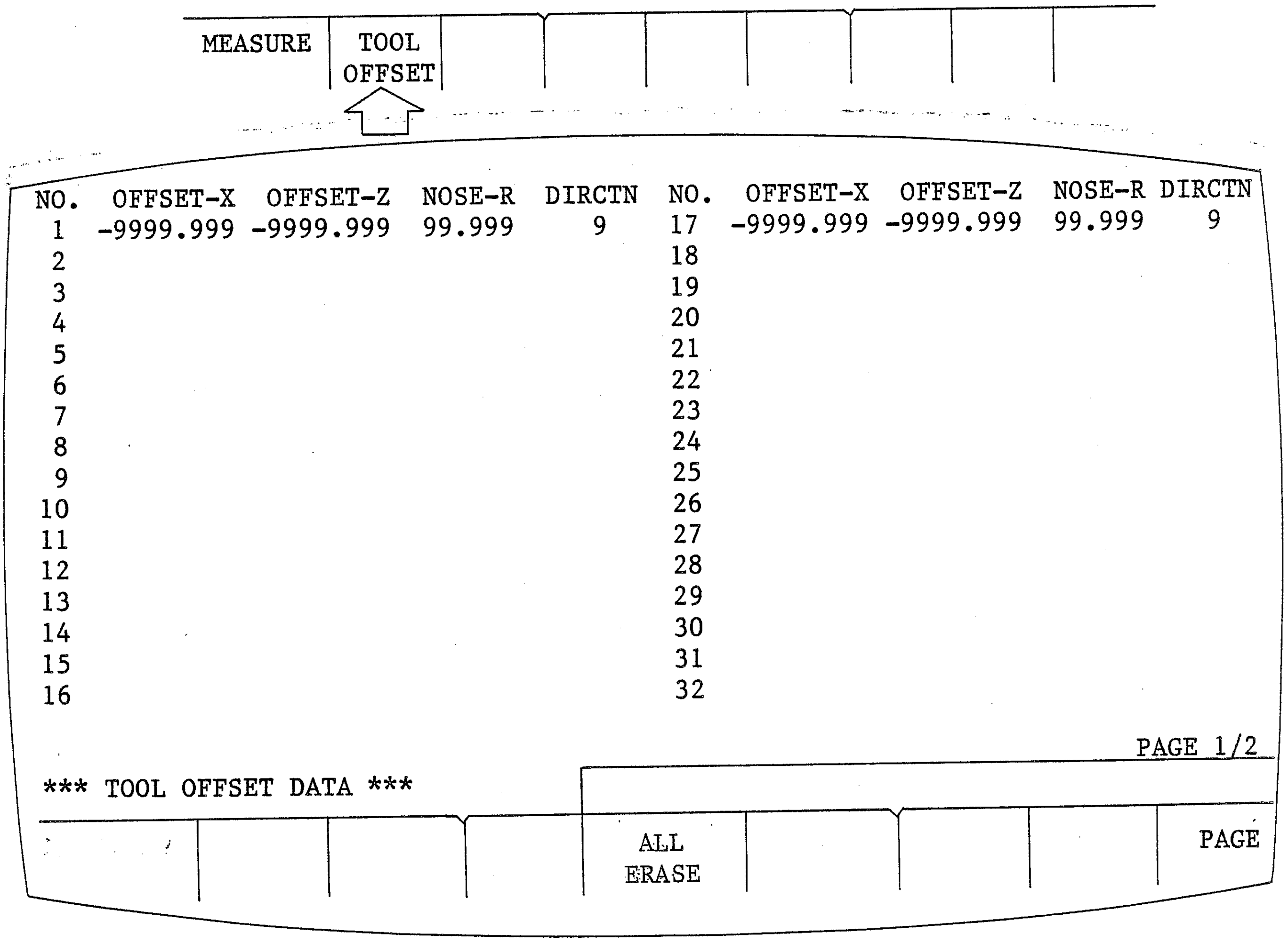
Note 2) Machining is not possible unless all the T-code tools to be used in EIA/ISO programs are registered in TOOL DATA display. They are registrable in TOOL DATA display using the tool registration function.

4.2 Tool Offset

Tool offset data, the values for the tool offset in EIA/ISO programs, are registered in TOOL OFFSET DATA display.
They are quite ineffective in MAZATROL programs.

4.2.1 Call of display

Press the (Display selection) key twice to call the TOOL SET menu. Then, press the TOOL OFFSET menu key to call the TOOL OFFSET DATA display.



```
NO.  OFFSET-X  OFFSET-Z  NOSE-R  DIRCTN  NO.  OFFSET-X  OFFSET-Z  NOSE-R DIRCTN
 1  -9999.999 -9999.999  99.999     9    17  -9999.999 -9999.999  99.999    9
 2                                       18
 3                                       19
 4                                       20
 5                                       21
 6                                       22
 7                                       23
 8                                       24
 9                                       25
10                                       26
11                                       27
12                                       28
13                                       29
14                                       30
15                                       31
16                                       32
                                                                      PAGE 1/2
*** TOOL OFFSET DATA ***
```

TOOL OFFSET DATA display

4.2.2 Registration of offset distance

(i) Enter data with the cursor transferred to the offset number to be registered.

Sixty four offset values are registrable on the maximum.

(1st page: offset Nos. 1 - 32
2nd page: offset Nos. 33 - 64)

(ii) Incremental input

Move the cursor to the offset number to be registered, and the INCRMENT menu will be displayed.

Enter data after inverting the INCRMENT menu by pressing its key. An input value will then be added to an already-registered value.

Note) "Z-OFFSET" in the PROGRAM FILE display is valid for G53 mode of EIA/ISO.

5. FEED FUNCTION

The following are the three tool feed rates:

(1) Rapid traverse rate

(2) Cutting feed rate

(3) Thread lead

5.1 Rapid Traverse Rate

Rapid traverse is generally used to move a tool during no-load operation (no machining is performed). The rapid traverse rate depends on the machine. Parameters (RFX, RFZ) are used for setting this rapid traverse rate.

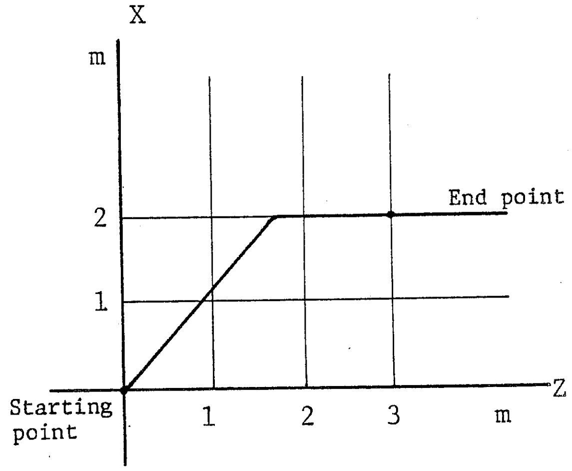
As the machine moves in each axis independently, the times in which it moves between the starting and end points of the axis are not equal.

For example, X- and Z-axis rapid traverse rates are 5 m/min and 8 m/min respectively and the following command is programmed,

G00U2000W3000;

(diameter programming)

the times in which the machine moves between starting and end points of X and Z axes are 12 sec and 22.5 sec, respectively.

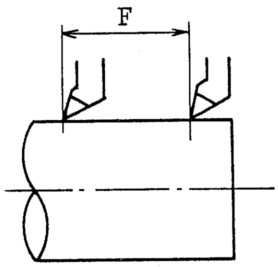


(Tool path in above example)

5.2 Cutting Feed Rate

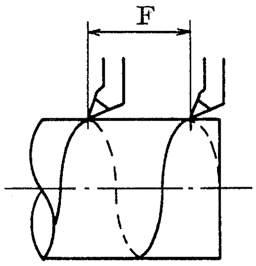
The tool feed rate during machining is called the cutting feed rate. The cutting feed rate is specified by the number following the address F. It is specified in feed per revolution or per minute.

Feed per minute
(mm/min, inch/min)



f = Tool feed distance
per minute

Feed per revolution
(mm/rev, inch/rev)



F = Tool feed distance
per revolution

		Feed per minute	Feed per revolution
Meaning		Tool feed distance per minute	Tool feed distance per revolution
Programming address		F	F
Setting G code		G98	G99
Range	Input in metric	1 - 15,000 mm/min (Fl - F15000)	0.01 - 500.00 mm/rev (Fl - F50000)
	Input in inch	0.01 - 600.00 inch/min (Fl - F60000)	0.0001 - 50.0000 inch/rev (Fl - F500000)

Note) G98 and G99 are modal. If one of these commands is programmed, it is effective until the another command of the same group is programmed.

5.3 Thread Lead

In thread cutting, the lead is commanded by the number following address F or E. The thread cutting is commanded by G32, G76 or G92.

G code	Meaning
G32	Thread cutting
G92	Thread cutting cycle (Canned cycle)
G76	Thread cutting cycle (Multiple repetitive cycle)

The range of lead length is as follows:

Address	Input in metric	Input in inch
F	0.01 - 500.00 mm	0.0001 - 50.0000 inch
E	0.0001 - 500.0000 mm	0.000001 - 9.999999 inch

Spindle speed is limited as follows:

$$N \leqq \frac{\text{Max. feed rate}}{\text{Lead length of thread}}$$

N: Spindle speed (rpm)

Lead: mm or inch (Max. 500 mm/rev or 50 inch/rev)

Max. commanded feed rates in feed per minute and per revolution are limited by motor or machine capacity.

Address F for thread lead is common to that for feed per minute and feed per revolution.

Address E is used only for thread lead length.

Either E code or F code which is specified later is effective for thread lead. For feed per minute and feed per revolution, existing F code is effective regardless of E code. Therefore, it is useful if the E code is used for thread lead specification and the F code for feed per minute and feed per revolution specifications because it is unnecessary to recommand the feedrate after thread cutting.

6. SPINDLE FUNCTION (S FUNCTION), TOOL FUNCTION (T FUNCTION) MISCELLANEOUS FUNCTION (M FUNCTION)

If addresses S, T and M are followed by numbers, other than the machine axial motion is controlled in three ways.

(1) Spindle function (S)
Controls spindle speed.

(2) Tool function (T)
Selects the tool and tool offset number to be used for machining.

(3) Miscellaneous function (M)
Performs various controls to stop the spindle or the program.
The following are the description of the above-mentioned functions.

6.1 Spindle Function (S Function)

(1) S 4-digit
The spindle speed (rpm) is directly commanded by address S and the following 4-digit number.

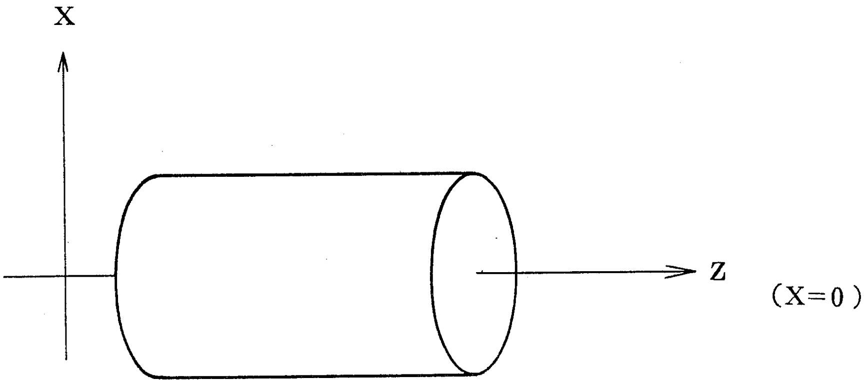
(2) Constant surface speed control
If surface speed (relative speed between tool and workpiece) is set after address S, the spindle speed is calculated so that the surface speed is always the specified value according to the instantaneous tool position.
The unit of surface speed is as follows:

Input unit	Surface speed unit
metric	m/min
inch	feet/min

To perform this control, the following G codes have to be set.

G code	Meaning	Unit
G96	Constant surface control	m/min feet/min
G97	Specifies the spindle speed	rpm

When this control is in effect, the work coordinate system must be set so that the rotary axis may be set to the Z axis (X = 0).



S (surface speed) in G96 mode is regarded as S = 0 until M03 or M04 is commanded.

(3) Clamp of the maximum spindle speed

The spindle speed automatically varies when surface speed is constantly controlled.

Fix the maximum spindle speed except that an increase of revolutions is allowed up to the machine capacity.

This instruction uses numbers following G50.

Example: G50 800; G50 S0;

When the spindle in constant surface speed control tries to exceed the value specified in the above program, the spindle speed is clamped at 800 rpm.

When S0 is given, the spindle speed is clamped at 0 rpm.

The spindle speed is also clamped in other than constant surface speed control mode.

Note 1) When positioning (G00), the surface speed is calculated on the basis of the X-coordinate value of end point.

Note 2) The speed specified as the constant surface speed in the control mode corresponds to the machining program tool moving instruction value, but not to the position obtained by adding the offset distance.

Note 3) Constant surface speed control is also available for thread cutting. Generally, use G97 to deactivate it before entering the thread cutting mode.

Note 4) Unless spindle speed, S, is specified for the block for which G97 is specified when changing to the constant surface speed invalid position from the constant surface speed mode, the final spindle speed in the later is used as that in the G97 mode.
The G96 mode S-value is used when changing to the constant surface speed control mode from the G97 mode.
Unless an S-value is specified, however, S = 0 m/min (feet/min).

Note 5) Leading zero can be omitted.

Note 6) When changing from the G96 to G97 modes, machining is performed with the last G96 revolutions unless an A-command is given in the G97 block.

```
Example)    N1  G97 S500; (500 rpm)
                 |
                 |
                 |
            N11 G96 S150; (150 m/min)
                 |
                 |
                 |
            N21 G97; <------ Machining is performed
                             before this block.
```

In this case spindle speed does not vary.

Note 7) When changing from the G97 to G96 modes, the S-value in the S-mode becomes effective.

```
Example 1)  N1  G96 S150;  (150 m/min)
                 |
                 |
            N11 G96 S500;  (500 rpm)
                 |
                 |
            N21 G96; <---- The command S150 at N1
                           becomes valid.
                           (150 m/min)
```

S=0 machining will be performed unless an S-command is input in the G96 mode.

```
Example 2)  N1  G96;        (0 m/min)
                 |
                 |
            N11 G97 S500;   (500 rpm)
                 |
                 |
            N21 G96;        (0 m/min)
```

6.2 Tool Function (T Function)

The number following the address T gives the tool selection instruction. A part of the number is used for the offset number for specifying the compensation amount of the tool offset.

Form of T-function input:

$T\underbrace{\square\square}_{\text{Tool number}}\underbrace{\square\square}_{\text{Tool offset number}}$

No leading zero is omitted.

Example: (for special models)

$T\square\square\underbrace{0\ 0}_{\text{(Note 1)}}\quad A\ \underbrace{\triangle\triangle}_{\text{(Note 2)}}\quad B\ \underbrace{✡✡}_{\text{(Note 3)}}$; Necessary when changing tools by the ATC.

T □□ ◇◇; Offset input

Tool offset number

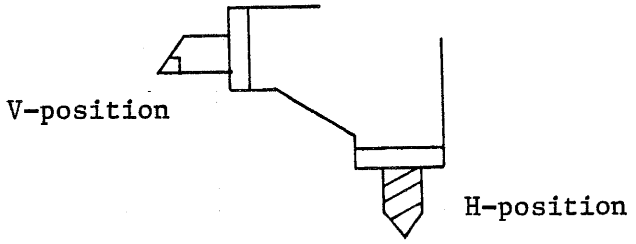
Note 1) Always cancel the tool offset by setting its number to "00" for the blocks with the above input form.

Note 2) Input of return tool number (1 - 64)
If no "A" data is inputted or when "A00" is set, the NC will automatically specify the tool to be returned.

Note 3) Input of next-time tool number (1 - 64)

Note 4) When programming tool commands using the V/H type ATC the following conditions should be considered.

When using 30-piece specification, input T0100 to T3000 and T3100 to T6000 commands for horizontal and vertical tools. POSITION COMMAND display indicates tool numbers 1 to 30, '1 and '30 for H- and V-position as in MAZATROL programs.



Various modes of use:

(1) When T-code is specified separately
When $\boxed{A6}$=1, the tool travels taking the compensation amount into account at the next travel instruction.
The tool exchange instruction and the motion instruction for the offset distance corresponding to tool offset number are executed.
Travel speed: rapid traverse or cutting feed is performed in G00 or other mode, respectively.

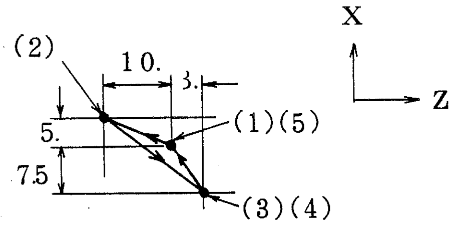
Examples:

(1) T0100 - T01 selected, tool offset cancelled.
(2) T0101 - T01 maintained, tool offset 01 selected.
(3) T0102 - T01 maintained, change to tool offset 02.
(4) T0202 - Exchange with T02, tool offset 02 maintained.
(5) T0100 - Exchange with T01, tool offset cancelled.

Tool offset distance

	X	Z
01	+10.0	-10.0
02	-15.0	+ 3.0

Note) X: diameter instruction



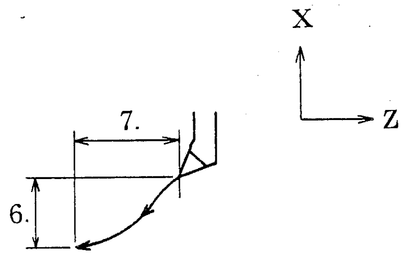
(2) If T code is specified for the same block as the motion instruction, priority is given to the motion instruction. The tool moves according to the motion instruction distance and the compensation amount set for the tool offset number, and the instruction tool number is selected.

Example: (T0100)

G00 U-10.0 W-10.0 T0101;

Tool offset distance

	X	Z
01	-2.0	3.0



X axis ..... (-10.0) + (-2.0) = -12.0 (diameter)

Z axis ..... (-10.0) + (3.0) = - 7.0

Note) Refer to Chapter 8. COMPENSATION FUNCTION for details of the tool offset.

(3) Tool offset number: 0 - 64

6.3 Miscellaneous Function (M Function)

A 3-digit figure is commanded following address M. Usually, M codes are used for ON/OFF control of machine function.
Following M codes (M00, M02, M03, M98, M99, M198 and M199) are used for special meaning.

(1) M00: Program stop
Machine operation is stopped after a block containing M00 is executed. The operation can be restarted by pressing the ▭ (CYCLE START) key.

(2) M01: Optional stop
Similar to the M00, automatic operation is stopped after a block which contains M01 is executed. (Press the ▭ (CYCLE START) key for restart.) This code is effective only when the OPT. STOP menu has been set on.

(3) M02, M30: End of program
- i) It shows the end of main program. It is necessary for registration from tape to NC memory.
- ii) Automatic operation is stopped and the NC unit is cleared.
- iii) Only M30
  The NC tape is rewound back to the start of the program.

(4) M98: Call of subprogram
This code is used to enter a subprogram. Refer to Chapter 10. SUBPROGRAM for details.

(5) M99: End of subprogram
This code shows the end of a subprogram. Executing M99 take the control back to the main program. Refer to Chapter 10. SUBPROGRAM for details.

(6) M198 and M199: End of program 2

Should be used to call the next program. See "End Processing" for further details.

Note 1) Specify one M code per block.
If several M codes are specified for one block, only the last one is used.

Note 2) After running M00 and M01, the machine will stop and the □ (CYCLE START) light will remain lit.

Note 3) When an automatic machining mode program containing M02 is terminated, the program data can not be changed unless the // (RESET) key is pressed. If an attempt is made to alter program data, Alarm 437, "M02 STOP MODE", will be displayed.

Note 4) M98, M99, M198 and M199 are the M-codes for internal numerical controller processing and cannot be output.

Note 5) When automatic operation is terminated using M02, no related program can be run from the beginning unless the // (RESET) key is pressed. If the □ (CYCLE START) key is pressed instead, machining will continue from the next block until M02 is encountered.

Note 6) See the machine operating manual for entering M-code commands.

7. PREPARATORY FUNCTION (G FUNCTION)

Preparatory function is used for various machining modes that are carried out by tool transfer.

A 2-digit number following address G determines the meaning of the command of the block concerned.

The G codes are divided into the following two types.

Type	Meaning
One-shot G code (called unmodal)	The G code is effective only at the block in which it was specified.
Modal G code	The G code is effective until another G code in the same group is commanded.

G-code list for MAZATROL CAM T-2 is as follows:

G code	Group	Function
◤ G00	01	Positioning (rapid traverse)
◤ G01		Linear interpolation (cutting feed)
G02		Circular interpolation CW
G03		Circular interpolation CCW
G04	00	Dwell
G10		Offset value setting
G27	00	Zero point return check
G28		Return to zero point
G29		Return from zero point
G30		Return to fixed point
G31		Skip cutting
G32	01	Straight and taper thread cutting
G36	00	Setting the target value in the measurement
G37		Operation and compensation of measurement data
◤ G40	07	Tool nose radius compensation cancel
G41		Tool nose radius compensation left
G42		Tool nose radius compensation right

G code	Group	Function
G50	00	Setting of coordinate system, Setting of maximum spindle speed
G52		MAZATROL Coordinate system cancel
G53		MAZATROL Coordinate system selection
G68	13	Mirror image for double turrets ON
◤ G69		Mirror image for double turrets OFF
G70	00	Finishing cycle
G71		OD roughing cycle
G72		Face roughing cycle
G73		Closed loop cutting cycle
G74		Face cutting-off cycle
G75		OD cutting-off cycle
G76		Thread cutting cycle
G90	01	Cutting cycle A
G92		Thread cutting cycle
G94		Cutting cycle B
◤ G96	02	Constant surface speed control
◤ G97		Constant surface speed control cancel
◤ G98	05	Feed per minute
◤ G99		Feed per revolution

Note 1) The G codes marked with ◤ are initial G codes in each group. That is, when the power is turned on or when the ⧄ (RESET) key is pressed under the status in which the system parameter by which resetting initializes G codes is effective, those G codes are set. For G00 and G01, or G98 and G99, either of them is selected for the initial G codes by setting of parameters.

Note 2) The G codes in the group 00 are not modal. They are effective only in the block in which they are commanded.

Note 3) An alarm occurs when a G code not listed in the above table is commanded. When an optional G code not contained in the control is specified, an alarm occurs.

Note 4) A number of G codes can be commanded in a block even if they do not belong to the same group. When a number of G codes of the same group are specified, the G code specified later is effective.

Note 5) G codes depend on group numbers.

Note 6) If group 00 and other modal G-codes are specified for one block, the former is used, the latter is ignored. The modal G-codes are, however, renewed.

Note 7) For G codes for measuring (G31, G36 and G37), see "Instruction Manuals for Optional Functions".

7.1 Setting of Coordinate System (G50)

When it is desired to move a tool by absolute command, the coordinate system must be set in advance.

(1) Setting of work coordinate system

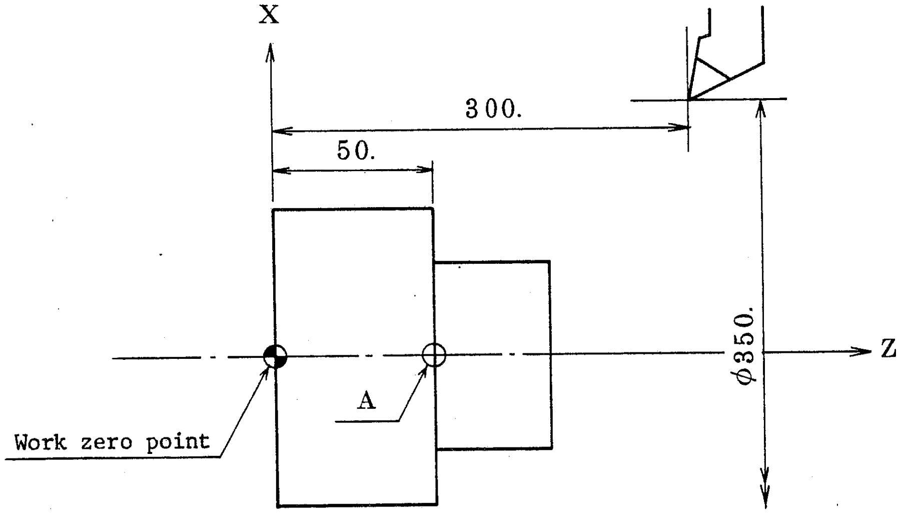
The coordinate system is established by the following command:

G50 X ___ Z ___ ;

This command creates a coordinate system in which the origin of the system is at the distances (x and z) specified in the command from the tool position. This coordinate system is called the work coordinate system. Once created, the subsequent absolute command refers to the coordinate value in this work coordinate system.

It is not necessary to give instructions to all axes at once. A change of coordinate system during a program is necessary only if an axis requires it.

Example 1: Tool nose position used for setting:

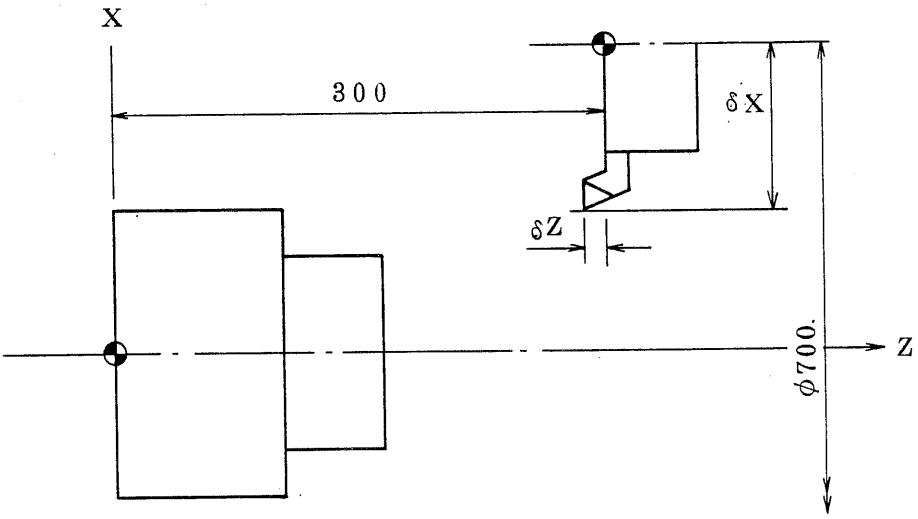


G50X350.0Z300.0;

Point A is used as the zero point for setting the coordinate system:

G50X350.0Z250.0;

Example 2: A reference point used for setting:



G50X700.0Z300.0;

This example uses the center of the rotation of turret. Any other points are used for reference. δX and δZ are handled in the tool offset mode.

Refer to 8.1 "Tool Offset" for details.

Note 1) When the coordinate system is set with a G50 command during offset mode, the coordinate system in which coordinate value of a tool excluding the offset value is the specified position, is established.

Note 2) The tool nose radius compensation is cancelled temporarily by G50 command.

Note 3) The S data specified for the same block as G50 differ from those of ordinary S in that they are used to set the spindle speed clamped.

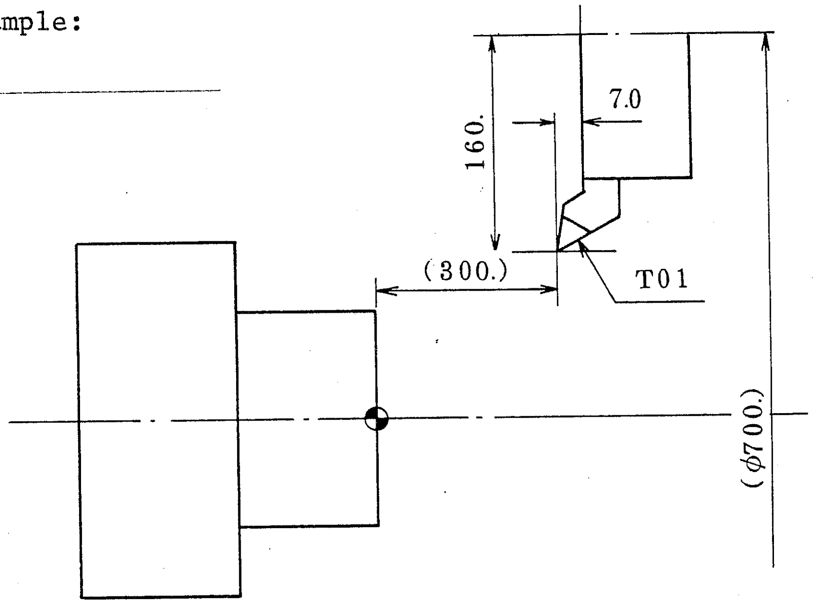
(2) Shift of work coordinate system

The work coordinate system can be shifted using the following command:

G50U$\underline{(u)}$ W$\underline{(w)}$;

The new, shifted work coordinate system in which the coordinate values of the tool nose are (x+u) and (z+w) is programmed against the present work coordinate system in which the coordinate values of it are x and z.

Example:



(G50X700.0Z300.0;)
(T01)
G50 U-320.0 W-7.0;

The work coordinate system with same zero point can be set by the above G50 command, when using a tool T01 as in the above figure.

(3) G50 command and current and machine position displays

Example 1) OFFSET DATA (-10.0, -10.0)

PROGRAM		PARAMETER (A6)=0		PARAMETER (A6)=1	
		Current position	Machine position	Current position	Machine position
N01	G28U0.W0.;	(0, 0)	(0, 0)	(0, 0)	(0, 0)
N02	G50X0.Z0.;	(0, 0)	(0, 0)	(0, 0)	(0, 0)
N03	T0101;	(0, 0)	(0, 0)	(-10, -10)	(-10, -10)
N04	G00X50.Z50.;	(40, 40)	(40, 40)	(40, 40)	(40, 40)
N05	G50X0.Z0.;	(-10, -10)	(40, 40)	(-10, -10)	(40, 40)
N06	G00X50.Z50.;	(40, 40)	(90, 90)	(40, 40)	(90, 90)
N07	T0100;	(40, 40)	(90, 90)	(50, 50)	(100, 100)
N08	G00X0.Z0.;	(0, 0)	(50, 50)	(0, 0)	(50, 50)
N09	G28U0W0;	(-50, -50)	(0, 0)	(-50, -50)	(0, 0)
N10	M02;	(,)	(,)	(,)	(,)

Example 2) OFFSET DATA (-10.0, -10.0)

PROGRAM		PARAMETER (A6)=0		PARAMETER (A6)=1	
		Current position	Machine position	Current position	Machine position
N01	G28U0.W0.;	(0, 0)	(0, 0)	(0, 0)	(0, 0)
N02	G50X0.Z0.;	(0, 0)	(0, 0)	(0, 0)	(0, 0)
N03	G00X50.Z50.;	(50, 50)	(50, 50)	(50, 50)	(50, 50)
N04	G50X0.Z0.T0101;	(0, 0)	(50, 50)	(0, 0)	(50, 50)
N05	G00X50.Z50.;	(40, 40)	(90, 90)	(40, 40)	(90, 90)
N06	T0100;	(40, 40)	(90, 90)	(50, 50)	(100, 100)
N07	G00X0.Z0.;	(0, 0)	(50, 50)	(0, 0)	(50, 50)
N08	G28U0W0;	(-50, -50)	(0, 0)	(-50, -50)	(0, 0)
N09	M02;	(-50, -50)	(0, 0)	(-50, -50)	(0, 0)

Example 3) OFFSET DATA (-10.0, -10.0)

PROGRAM		PARAMETER (A6)=0		PARAMETER (A6)=1	
		Current position	Machine position	Current position	Machine position
N01	G28U0.W0.;	(0, 0)	(0, 0)	(0, 0)	(0, 0)
N02	G50X0.Z0.;	(0, 0)	(0, 0)	(0, 0)	(0, 0)
N03	G00X50.Z50.;	(50, 50)	(50, 50)	(50, 50)	(50, 50)
N04	G50X0.Z0.T0101;	(0, 0)	(50, 50)	(0, 0)	(50, 50)
N05	G00X50.Z50.;	(40, 40)	(90, 90)	(40, 40)	(90, 90)
N06	G28U0W0;	(-50, -50)	(0, 0)	(-50, -50)	(0, 0)
N07	M02;	(,)	(,)	(-60, -60)	(-10, -10)

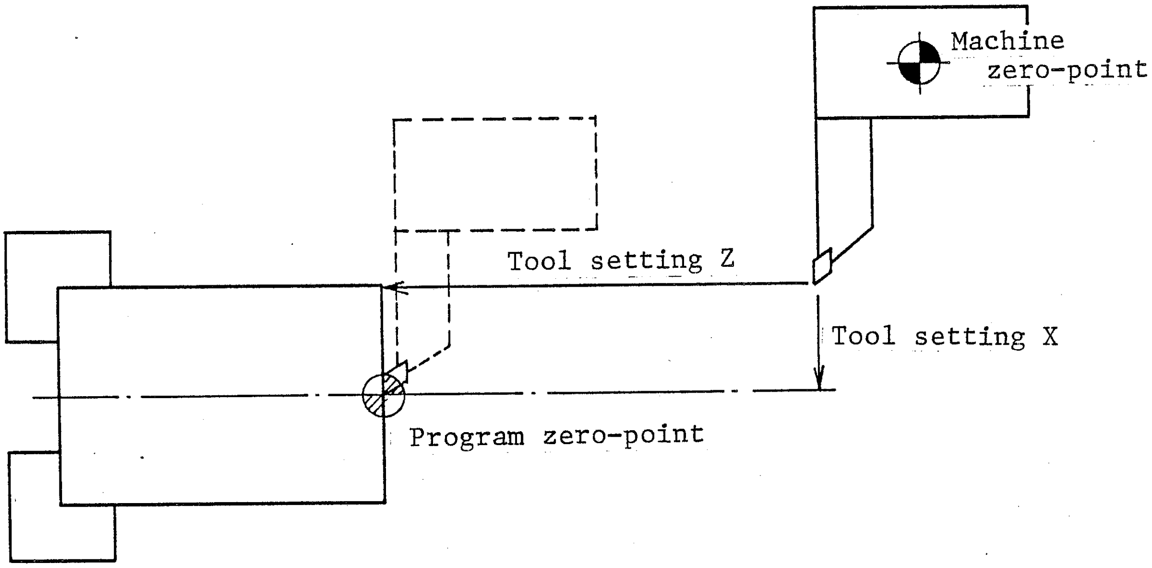
Note) Parameter A6 is used to determine whether an axis should be moved (=0/=1) using a separate offset command.

7.2 Entering MAZATROL Coordinate System (G53):

G-codes commands are used to change from G50 to the MAZATROL coordinate system

The MAZATROL coordinates specify the current tool tip position with respect to preset machine and tool positions.

See Tool Setting in the MAZATROL operating manual for further details. Tool setting principles are as follows:

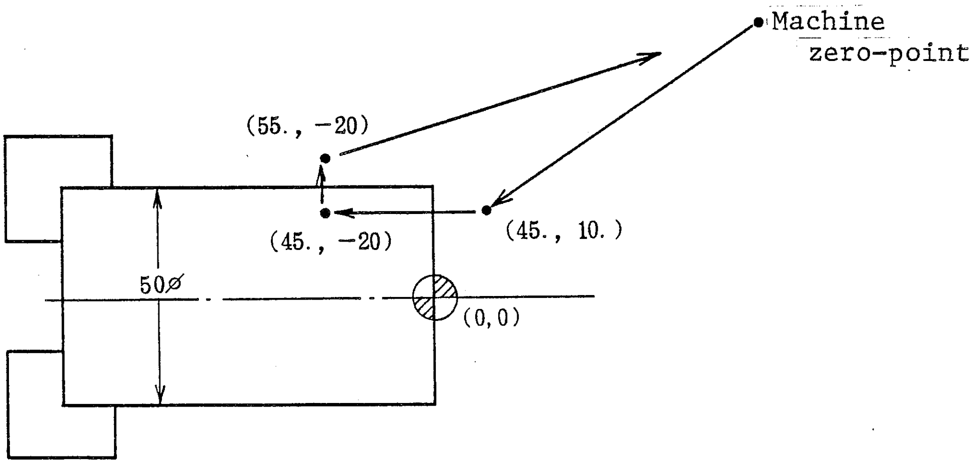


The position to which the tool tip is moved along the axis of rotation with respect to the work surface is called the program zero-point as illustrated above. It is equivarent to being designated as G50X0Z0; at this position.

Tool setting: the machine position error caused when a tool tip is moved to the program zero-point.

(The MAZATROL coordinate system makes it easier to use the program coordinates since all are tool tip positions. The tool is moved by the program prepared using the tool tip as a reference point based on the display data. The conventional G50 coordinate system requires many trial cuts while watching the machine position and using the tool offset data.)

Example)



Program producing the tool path illustrated above.

MAZATROL coordinate system ——→ (G0 X45.Z10.;)

```
G28U0W0;
T0100;
G53;
G0 X45.Z10.;
G1 Z-20.F10;
   X55;
G28U0W0;
```

1) Include the G53 command in single blocks whenever possible except when in the G53 mode.

2) If the G53 command is given in the G52 mode (ordinary G50 coordinate system mode), the current G50 will be temporarily cancelled and the coordinates for the current tool selected will be input.
At the same time, the current position counter resets to new coordinates.

3) If G53 is used in a block with other move commands the latter will be run and the coordinates will change. Independent of the location of G53 in the program.

4) If G53 and G50 commands are included in one block, they will be run in the same order regardless of the location of G53 in the program.

5) When the G53 command is run, an alarm will be displayed if all axes except C have been reset to zero without reaching the limit switches.

6) When tool change command (T) is run in the G53 mode, the MAZATROL tool coordinate system will be reloaded automatically.

7) When the G50 command is given in the G53 mode, a coordinate change is made using the $\boxed{\text{P11}}$ parameter.

8) Return to the G50 coordinate system, which has been temporarily cansellled using RESET or PROGRAM END (M30, M198, M199 and %) while in G53 mode, is possible using parameter $\boxed{\text{P11}}$.

Parameter $\boxed{\text{P11}}$:

Used when setting the range from 0 to 3.

G50 command in G53 mode Resetting or program end	Valid	Invalid (MAZATROL coordinate system)
No return to G50 coordinate system	P11 = 0	P11 = 1
Return to G50 coordinate system	P11 = 2	P11 = 3

Note) The G50 coordinate system is reactivated before changing to the G53 mode.

When the G50 command is not valid ($\boxed{\text{P11}}$ = 1,3), no coordinates are changed unless G50 S__, (turning clamp command) is given.

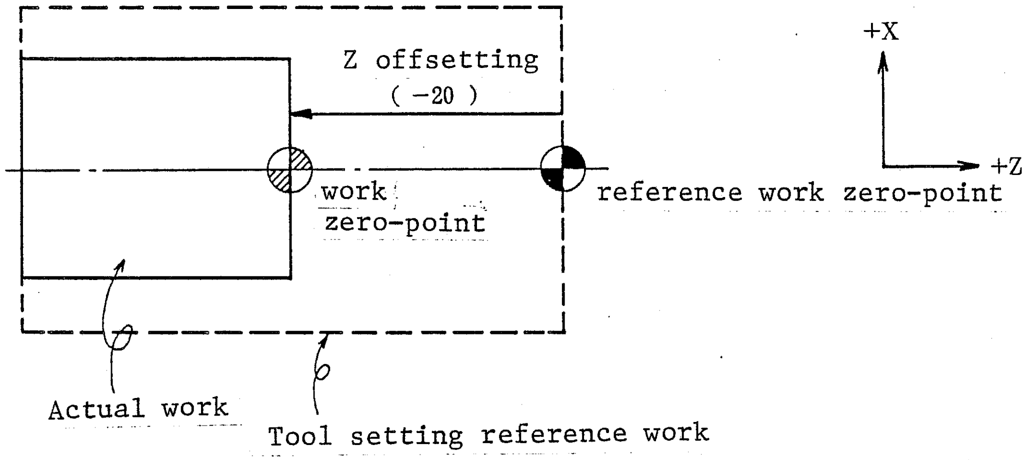
Z offsetting:

For offsetting the tool along the Z-axis with respect to the zero-point, see the MAZATROL operating manual.

For machining work pieces with different lengths, it is necessary to establish the Z-axis distance of the zero-point (work zero-point) from the tool zero-point (reference work zero-point).

Since it is difficult to reset the tool for each work piece, do not change the tool zero point after it has been set. Instead set the new coordinates for each work piece by Z-offsetting.

Example) Work piece No.1



If the actual work zero-point is 20mm along the Z-axis from the reference zero point as illustrated above, enter -20 in PROGRAM FILE picture Z-OFFSET space for work piece No.1. The coordinates will be moved in the Z direction. When X0 and Z0 commands are in the program, tool tip will move to the new zero-point.

Use the G52 command to cancel the G53 (MAZATROL coordinate system setting) mode.

1) Any G52 command in the G52 mode will be ignored.

2) If G52 command is given in the G53 mode, the current MAZATROL coordinate system will be replaced by the G50 system automatically.
The current position counter indication changes from G53 to G52.

3) If G52 and other move commands are given to one block, the latter is run and the coordinates will be changed irrespective of the position of G52 in the program.

4) If G52 and G50 commands are input to the same block, they will be run in this order.

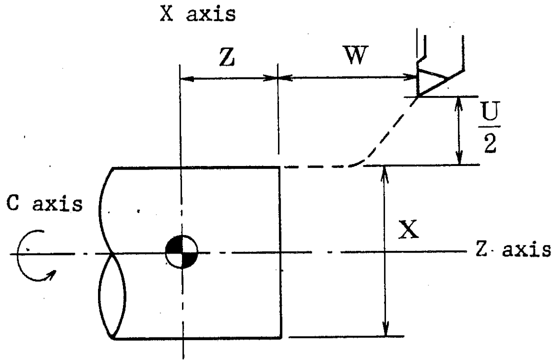
5) Use the G52 command for single blocks if possible.

7.3 Positioning (G00)

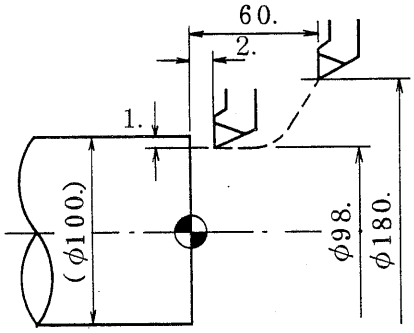
G00 specifies positioning.

A tool moves to the (X, Z) position in the work coordinate system or from it's current position to the position specified as the (U, W) distance at the rapid traverse rate in each axis independently. The tool path to a specified end point is not always linear.

G00X(U) —— Z(W);



Example:



G00 X98.0 Z2.0;

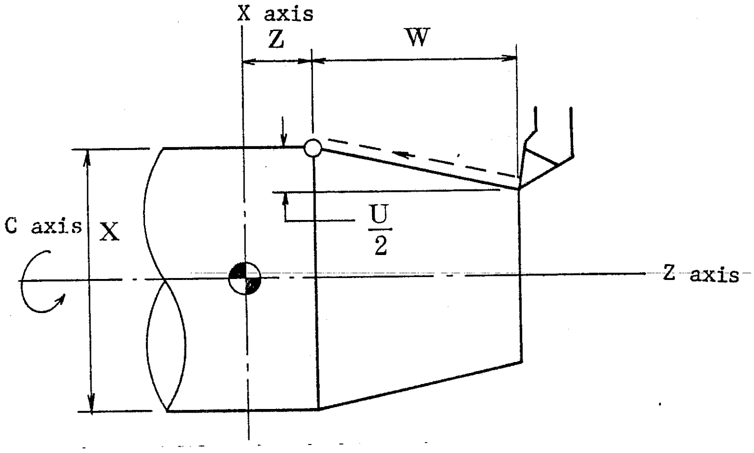
or G00 U-82.0 W-58.0;

Note) The rapid traverse rate in the G00 command is set for each axis independently by the parameter. Accordingly, rapid traverse rate cannot be specified by the address F.

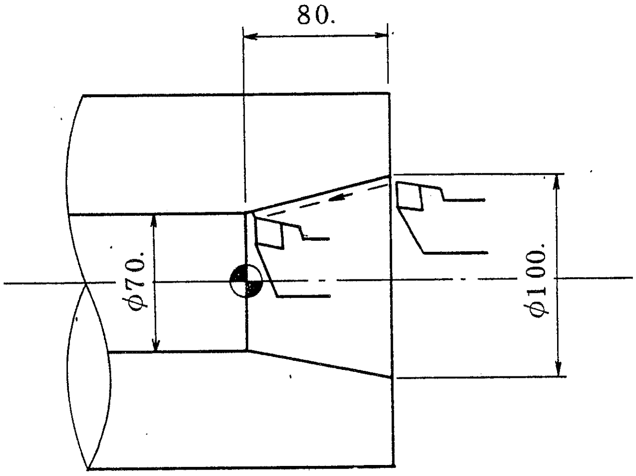
7.4 Linear Interpolation (G01)

Linear interpolation moves a tool linearly to the (X, Z) position in the work coordinate system or from it's current position to the position specified by the (U, W) values at the feedrate commanded by address F. Linear interpolation is specified by G01.

```
G01X(U) __ Z(W) __ F __;
```



Example:



```
G01 X70.0 Z0. F0.5;
or G01 U-30.0 W-80.0 F0.5;
```

Note) The following are axis travel speeds:

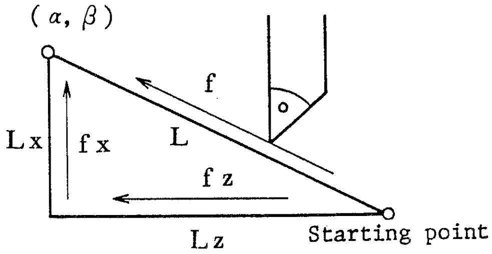
Example: G01 X Z Ff;

X-axis speed: $fx = \frac{Lx}{L} \cdot f$

Z-axis speed: $fz = \frac{Lz}{L} \cdot f$

$L = \sqrt{Lx^2 + Lz^2}$

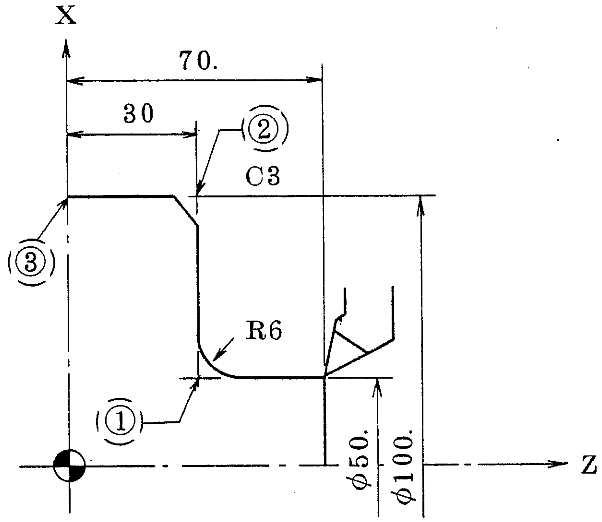
$f = \sqrt{fx^2 + fz^2}$



7.5 Chamfering and Corner R

A chamfering or corner arc can be inserted between two blocks specified by the linear interpolation (G01).

I and K always specify radius value.



```
(X50.0 Z70.0) ... Starting point
G01 Z30.0 R6.0 F__; ... (1)
X100.0 K-3.0 ; ........ (2)
Z0 ................... (3)
```

Note 1) The movement for chamfering or corner R must be that of either an X or Z axis single movement in G01 mode. And the next block must be either and X or Z axis single movement perpendicular to the former block axis.

Note 2) The end point for the preceding block serves as the starting point of the next block.

Example: G01Z270.0R6;
X860.0K-3; Starting point of this block: Z270.0

Note 3) The following commands generate alarms.

1) Both X and Z axes were commanded and one of either I, K or R was commanded in G01 mode.

2) Two of the following I, K and R were commanded in the same block in G01 mode.

3) Both X and I or Z and K were commanded in G01 mode.

4) Move distance along X or Z axis is smaller than chamfering or corner R value in th block in which chamfering or corner R was specified.

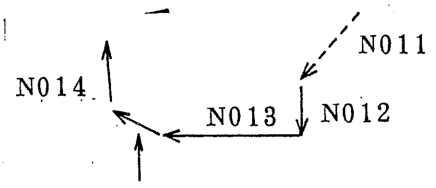
5) The block after the block which specified chamfering or corner R is not perpendicular to the former block in G01 mode.

Note 4) Chamfering and corner R cannot be used in a thread cutting block.

Note 5) Even if executed with SINGLE STEP operation, both chamfering and coner R are carried out in sequence.

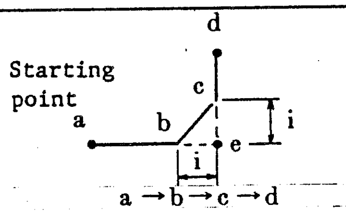
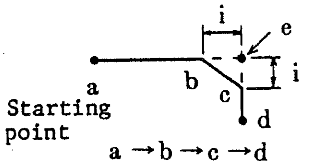
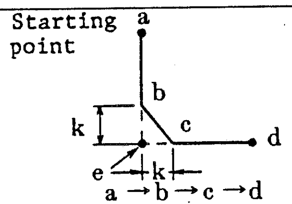
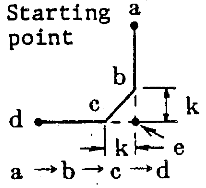
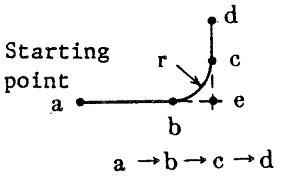
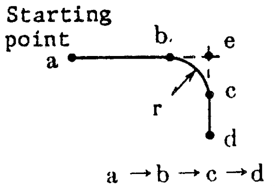
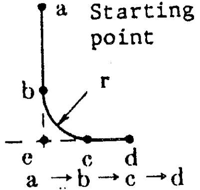
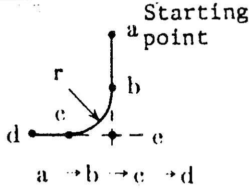
Note 6) Note the execution point when the M and T commands are given to the same block.

```
N011    G00 X100.Z0;
N012    G01 X90.F0.5;
N013    Z-20.R0.5 M08;
N014    X100.
```



M08 execution point

Command format of chamfering and corner R

Item	Command	Tool movement	Remarks
Chamfering Z → X	G01Z(W)e Ii; X(U)d; .. Next block		Specifies a movement to point e. In the next block, a X-axis movement is commanded with d ≧ e + 2i.
	G01Z(W)e I-i; X(U)d; .. Next block		Specifies a movement to point e. In the next block, a X-axis movement is commanded with d ≦ e - 2i.
Chamfering X → Z	G01X(U)e Kk; Z(W)d; .. Next block		Specifies a movement to point e. In the next block, a Z-axis movement is commanded with d ≧ e + k.
	G01X(U)e K-k; Z(W)d; .. Next block		Specifies a movement to point e. In the next block, a Z-axis movement is commanded with d ≦ e - k.
Corner R Z → X	G01Z(W)e Rr; X(U)d; .. Next block		Specifies a movement to point e. In the next block, a X-axis movement is commanded with d≧ e + 2r.
	G01Z(W)e R-r; X(U)d; .. Next block		Specifies a movement to point e. In the next block, a X-axis movement is commanded with d ≦ e - 2r.
Corner R X → Z	G01X(U)e Rr; Z(W))d; .. Next block		Specifies a movement to point e. In the next block, a Z-axis movement is commanded with d ≧ e + r.
	G01X(U)e R-r; Z(W))d; .. Next block		Specifies a movement to point e. In the next block, a Z-axis movement is commanded with d ≦ e - r.

7.6 Circular Interpolation (G02, G03)

Circular interpolation moves a tool along an arc specified in terms of the distance (I, K) between the starting point and the arc center or by the arc radius, to a point (X, Z) in the work coordinate system or from it's current position to the position specified by the (U, W) values. The feedrate is commanded by F code.

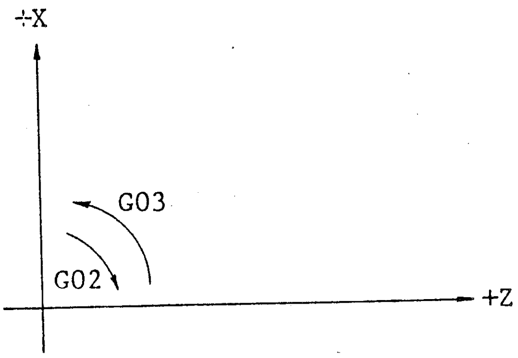
Operation is performed in accordance with the following command format:

$$\begin{Bmatrix} \text{G02} \\ \text{G03} \end{Bmatrix} \text{X(U)}\_\_\_ \text{Z(W)}\_\_\_ \begin{Bmatrix} \text{R}\_\_\_ \\ \text{I}\_\_\_ \text{K}\_\_\_ \end{Bmatrix} \text{F}\_\_\_;$$

The meanings of the above are tabulated below in detail.

No.	Data to be given	Command	Meaning
1	Rotation direction	G02	Clockwise direction (CW)
		G03	Counterclockwise direction (CCW)
2	End point position	X, Z	End point position (X, Z) in the work coordinate system
	Distance to the end point	U, W	Distance from starting point to end point
3	Distance from starting point to center	I, K	Distance from starting point to center (Always radius value)
	Arc radius	R	Arc radius. However arc is up to 180 degrees.

In the right-hand coordinate system, clockwise and counter-clockwise directions are as follows:

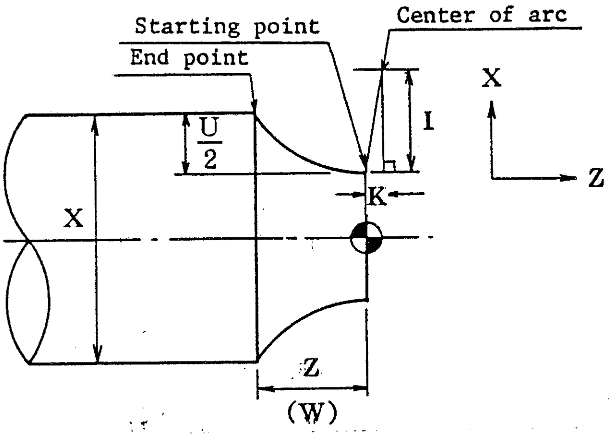


The command is given in the following format:

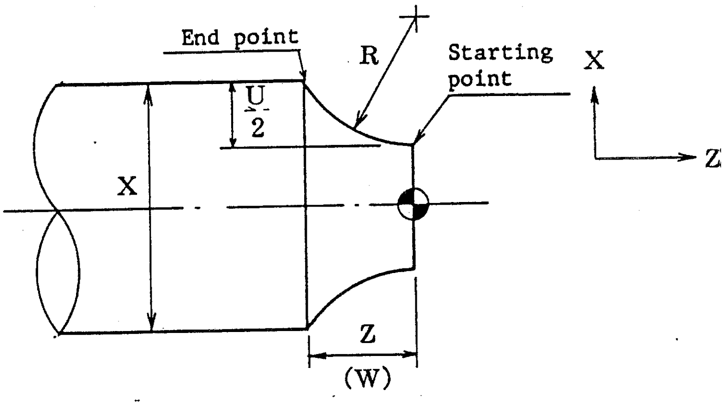
1) G02 mode

```
G02X(U)___Z(W)___I___K___F___; ..... I.K command
  or
G02X(U)___Z(W)___R___F___; .......... R command
```

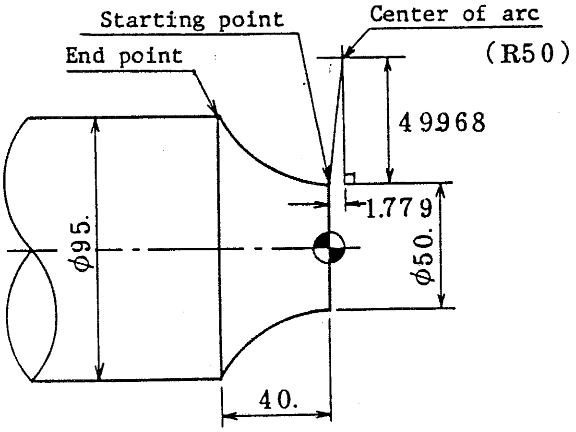
I.K command



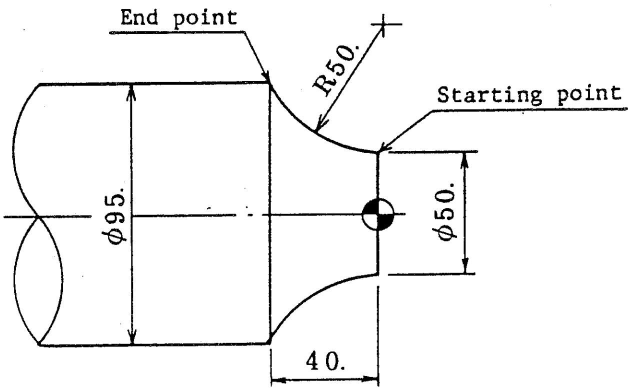
R command



Example 1:



Example 2:



```
(G01X50.0Z0;)
G02X95.0Z-40.0I49.968K1.779;
```

Therefore, R $=\sqrt{I^2 + K^2}$

```
(G01X50.0Z0;)
G02X95.0Z-40.0R50.0;
```

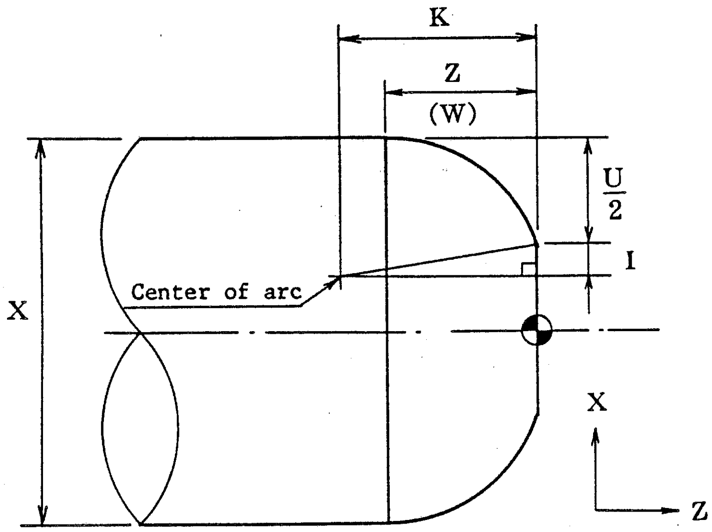
2) G03 mode

G03X(U)___Z(W)__I__K___F___; ..... I.K command

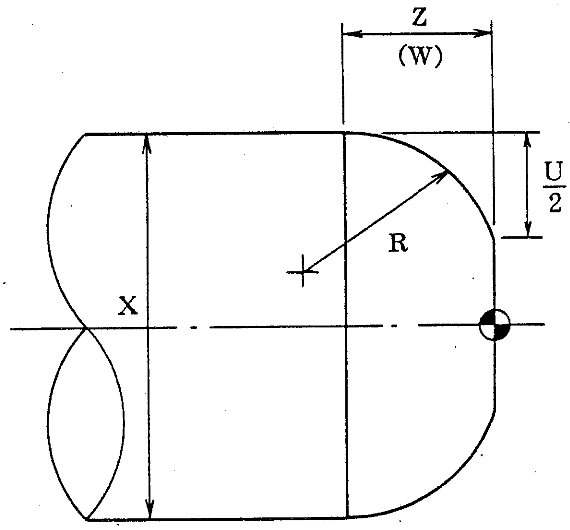
or

G03X(U)___Z(W)__R__F___; .... R command

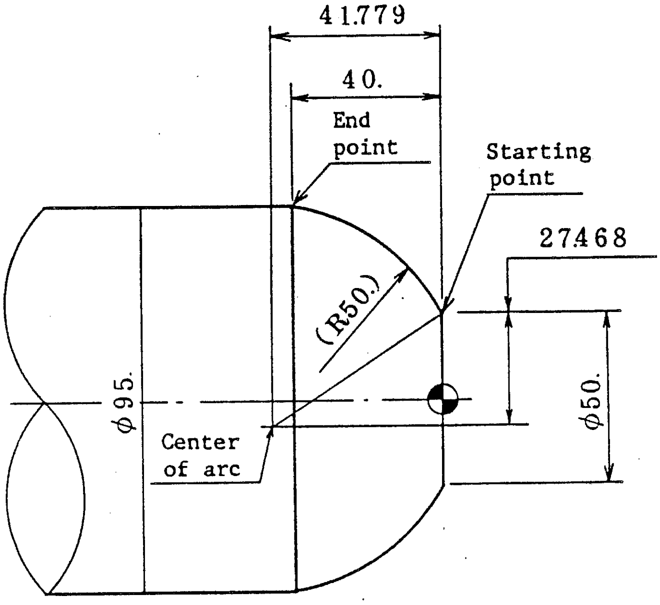
I.K command



R command



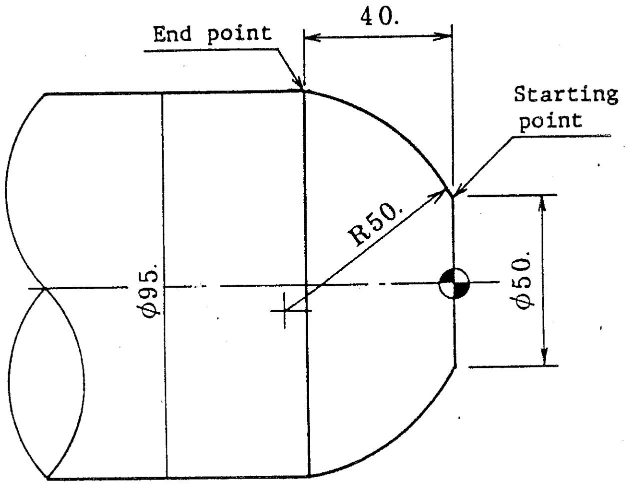
Example 1:



(G01X50.0Z0;)

G03X95.0Z-40.0I27.468K-41.779;

Example 2:



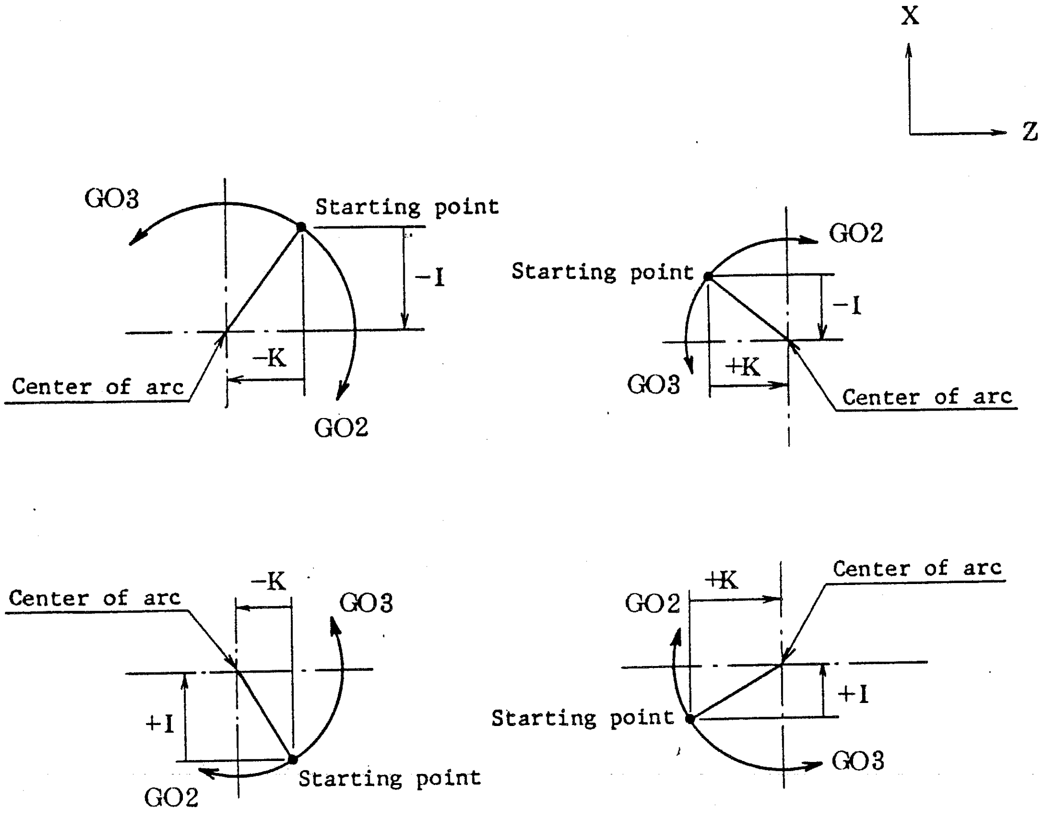
(G01X50.0Z0;)

G03X95.0Z-40.0R50.0;

Note 1) I and K are incremental values.
When either I or K is zero, the word can be omitted.

Note 2) I and K specify th distance from the starting point to the arc center along the X and Z axes respectively and their signs must be considered.

Relationship between starting point and signs of addresses I and K



Note 3) In either of the cases when all of X, Z, U and W are omitted, or when both U and W are specified to be zero or when X and Z are specified as the same position as the starting point, and when the arc center is programmed using addresses I and K, a 360 degrees circle (full circle) is assumed to be commanded.

Example: G02I ——— ;

Note 4) When address R is used, an arc less than 180 degrees is assumed to be commanded, so an arc more than 180 degrees cannot be commanded.

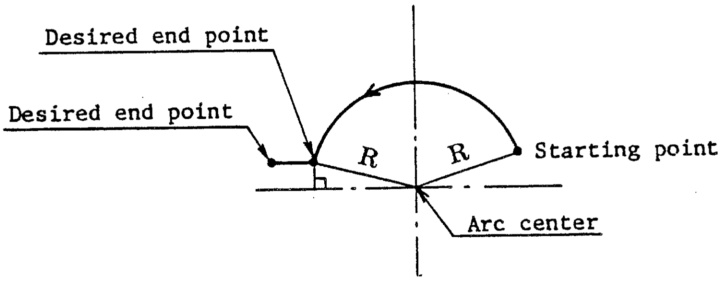
Accordingly: G02R —— ; and G03R —— ;

are not full circle but a circle of zero degree is assumed. So the tool does not move.

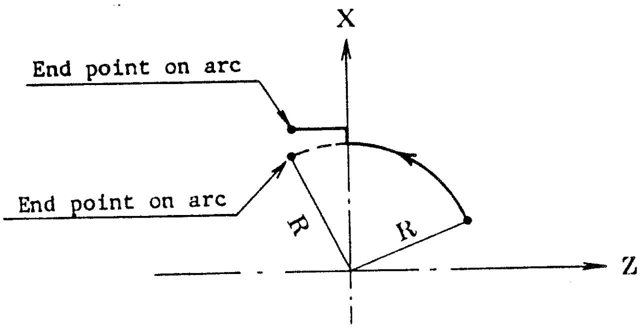
Note 5) When address R specifies a value less than half of the distance between the starting point and the end point, R is ignored and a half circle will be generated.

Note 6) When th end point is not on the arc, the tool moves as in the following figures.

(i) After a tool has reached the coordinate value of the end point in one axis, it moves to the coordinate value of the end point in the other axis.



(ii) When the arc does not coincide with a coordinate value of the end point, the following results.



Note 7) Feedrate in circular interpolation is specified with an address F (cutting feedrate). Error of the feedrate commanded against actual feedrate is within ±2%. However, this feedrate is the feedrate along an arc after tool nose radius compensation has been applied.

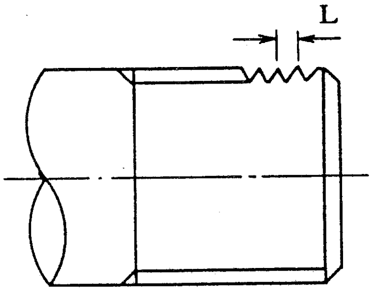
Note 8) When I, K and R are commanded simultaneously, R is effective and I and K are ignored.

Note 9) When both I and K are commanded as zero, the tool moves from starting point to end point linearly as in G01. As is also the case when R is commanded as zero.

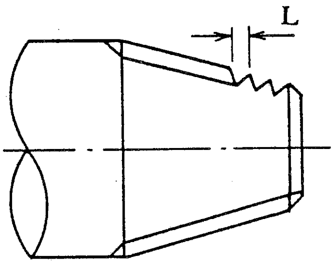
7.7 Thread Cutting (G32)

(1) Straight, taper and face thread cutting (G32)

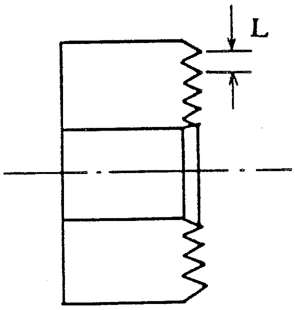
Taper and face threads in addition to straight threads can be cut by the use of a G32 command.



Straight thread



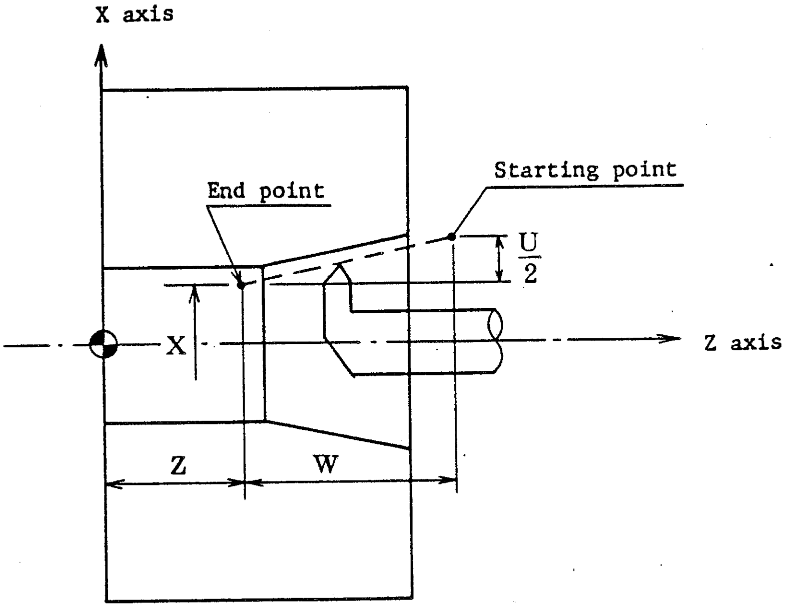
Taper thread



Face thread

The command shown below enables thread cutting with the lead specified by numeric values following address F or E.
The end point is specified by coordinate position (X, Z) in the work coordinate system or by move distance (U, W) from starting point to end point.

(G00 X(U) —— Z(W) —— ;) .... Starting point
G32 X(U) —— Z(W) —— F —— ; ..... End point
or G32 X(U) —— Z(W) —— E —— ; ..... End point

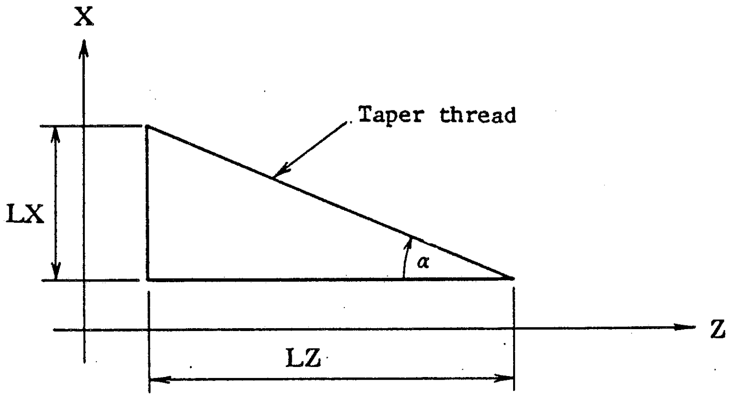


Note 1) F or E is used for as a lead instruction.
(Lead = pitch x number of threads)

Note 2) X(U) instruction can be omitted for straight thread.
Z(W) instruction can be omitted for face thread.

In general, thread cutting is repeated on the same tool path, in the course of rough cutting and finish cutting for a screw. Accordingly, the thread cutting is started at a fixed point and the tool path on the work is unchanged for repeated thread cutting. Note that the spindle speed must remain constant through the rough cutting to the finish cutting. If not, incorrect thread lead will occur.

In taper thread cutting, the lead whose value is greater either in the X or Z direction must be commanded.



If $\alpha \leqq 45°$ Lead is LZ

If $\alpha \geqq 45°$ Lead is LX

Thread lead must be specified by radius value.
In general, incorrect leads will be produced somewhat at the starting and end points of thread cutting. Considering this fact, the thread length should be commanded longer, to some extent, than required thread length.

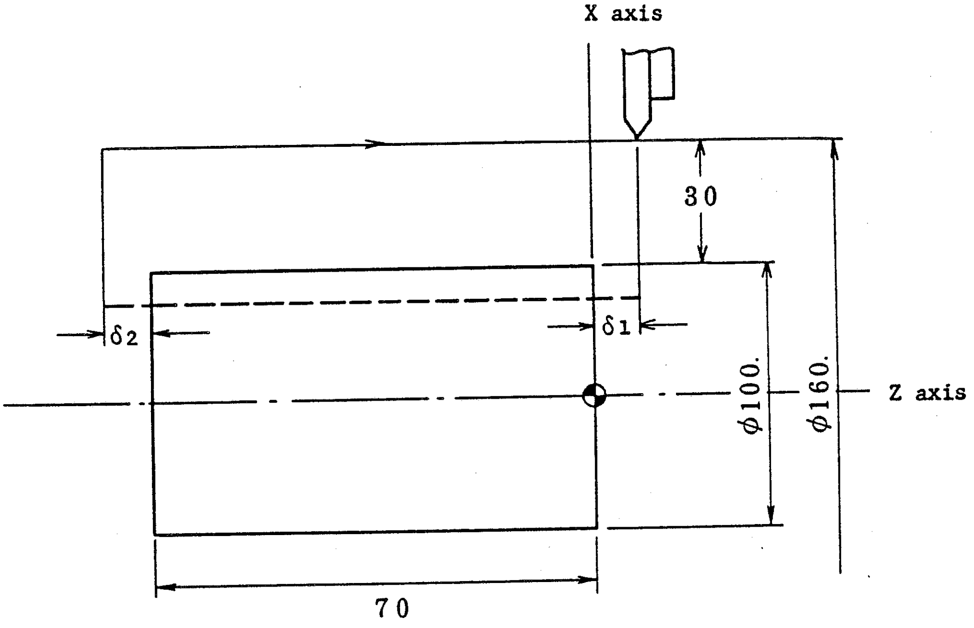
Example: Command value and thread lead ... F code designation

Input unit	F200	F1234
mm	2.0 mm	12.34 mm
Inch	0.02 inch	0.1234 inch

Example: Command value and thread lead ... E code designation

Input unit	E1234	E123456
mm	0.1234 mm	12.3456 mm
Inch	0.001234 inch	0.123456 inch

Example: (1) Straight thread cutting



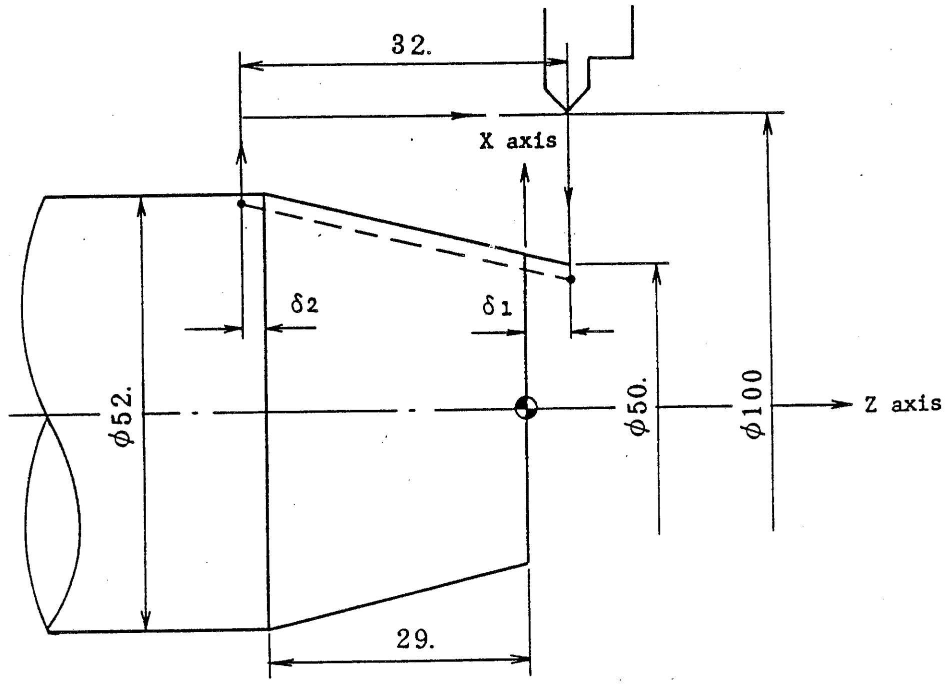
Programming is made with the following values:

Thread lead : 4 mm
δ_1 : 3 mm (Incomplete thread)
δ_2 : 1.5 mm (Incomplete thread)
Cutting depth: 1 mm (Cut twice)

(Milli-input, diameter programming)

```
G00 U-62.0;                                or G00 X 98.0;
G32 W-74.5 F4.0;                              G32 Z-71.5;
G00 U 62.0;                                   G00 X160.;
    W 74.5;                                       Z  3.;
    U-64.0; (For the second cut,                  X 96.;
            cut 1 mm more.)
G32 W-74.5;                                   G32 Z-71.5;
G00 U 64.0;                                   G00 X160.;
    W 74.5;                                       Z  3.;
```

Example: (2) Taper thread cutting



Programming is made with the following values:

Thread lead : 3.5 mm in the Z-axis direction

δ_1 : 2 mm

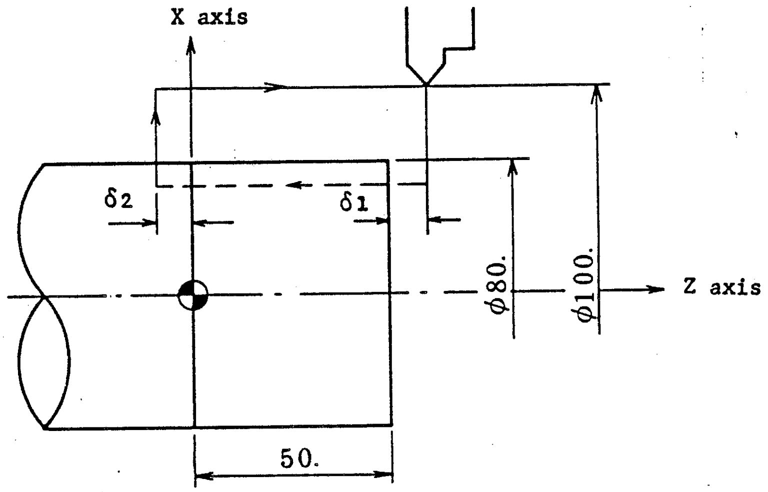
δ_2 : 1 mm

Cutting depth in X-axis direction is 1 mm (Cut twice).

```
(Milli-input, diameter programming)
G00 X 48.0;
G32 X 50.0 W-32.0 F3.5;
G00 X100.0;
    W 32.0;
    X 46.0; (For the second cut, cut 1 mm more.)
G32 X 48.0 W-32.0;
G00 X100.0;
    W 32.0;
```

Example: (3) Precision thread cutting

The high-accuracy machining instruction for such leads with fractions as specified in millimeters for inch threads can be given using a E code.



Programming is made with the following values:

Thread lead : 3 screw threads/inch (= 8.4666 ....)
δ_1: 3 mm
δ_2: 1.5 mm
Cutting depth: 1 mm (Cut twice)

(mm input, diameter programming)

```
G00 X 78.0;
G32 Z-51.5 E8.4667; (Round off to five decimal places.)
G00 X100.0;
    Z  3.0;
    X 76.0; (For the second cut, cut 1 mm more.)
G32 Z 51.5;
G00 X100.0;
    Z  3.0;
```

Note 1) E codes are effective only in thread cutting.

Note 2) If an F code is specified, the command value of the E code is ignored.

Note 3) Feedrate override is ineffective (fixed at 100%) during thread cutting.

Note 4) A parameter U19 setting selects whether dry run is effective or ineffective during thread cutting.

Note 5) When thread cutting is executed in the single block state, the tool stops after the execution of the first non-threading block (in which the tool moves) following that of the thread cutting block.

Note 6) When the mode is changed from automatic operation mode to manual operation mode during thread cutting, the tool stops after execution of chamfering from the thread cutting end point.

Note 7) When the previous block was thread cutting, cutting will be started immediately, without waiting for detection of the 1-turn signal even if the present block is for a thread cutting.

Note 8) When thread cutting blocks are consecutive, incorrect thread will occur for a few 10 msec between the two blocks. When cutting consecutive short screws, if buffering is not in time, incorrect thread will occur for a longer period of time.

Note 9) Because constant surface speed control is effective during face or taper thread cutting and spindle speed changes, correct thread lead may not be cut. Accordingly, constant surface speed control should not be used during thread cutting. G97 (constant surface speed cancel, constant spindle speed) must be commanded.

Note 10) The block before the thread cutting block must not include chamfering or corner R.

Note 11) The thread cutting block must not include chamfering or corner R.

Note 12) In thread cutting mode, the spindle speed override is effective. However, if the override is applied to the thread cutting, the override rate cannot be changed.

Note 13) "Thread cutting pause" function is not effective for G32.

7.8 Automatic Zero Point Return (G27 - G30)

(1) G27 Zero point return check

The zero point is a fixed point on the machine and the manual zero point return can move a tool to the zero point.
A G27 command confirms whether a tool has reached the zero point or not by the program which was made to return it to the zero point.

```
G27X(U) —— Z(W) —— ;
```

The tool moves to the commanded position at the rapid traverse rate by the above command. If the position which the tool has reached is the zero point, the HOME lamp is lighted. If only one axis has reached the zero point, the HOME lamp of that axis is lighted. If the commanded axis does not reach the zero point after positioning, an alarm is displayed.

Note 1) The position commanded by the G27 command will shift by the offset value if an offset has been specified. Accordingly, unless a tool which has been offset reaches the zero point, the lamp is not lighted. Normally, offset should be cancelled before a G27 is specified.

(2) G28 Automatic return to zero point

G28X(U) —— Z(W) —— ;

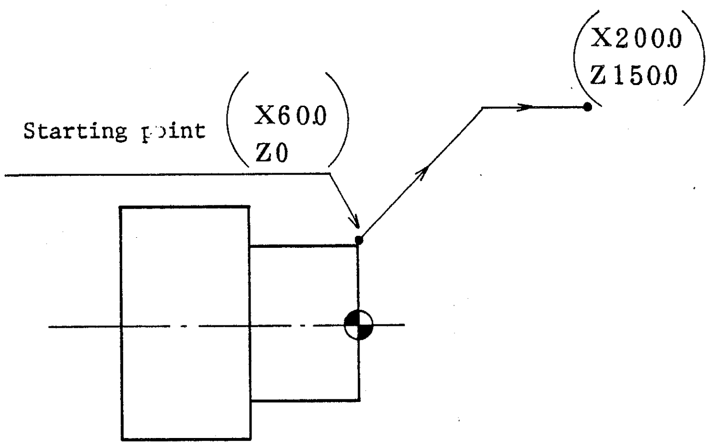
This command provides an automatic return to zero point for commanded axes. The coordinate values of all axes at the end point of the move command for this block are stored. This point is called the intermediate point for zero point return. The G28 block functions to position all commanded axes to the intermediate point at rapid traverse rate, the commanded axes are moved to the zero point at rapid traverse speed, and if the machine lock has not been effected, the HOME lamp lights up.

The positioning to intermediate point or to zero point is equivalent to the positioning by G00.

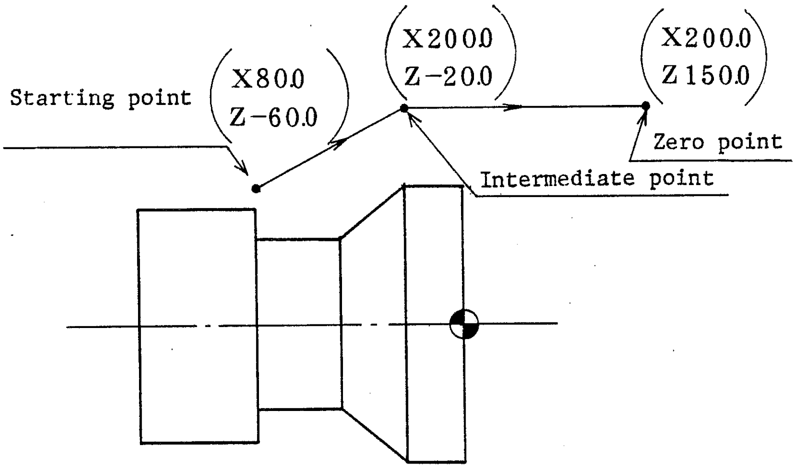
In general, this command is used to change tools.

Therefore, in principle, it is advisable, for safety, to cancel the tool nose radius compensation and tool offset before executing this command.

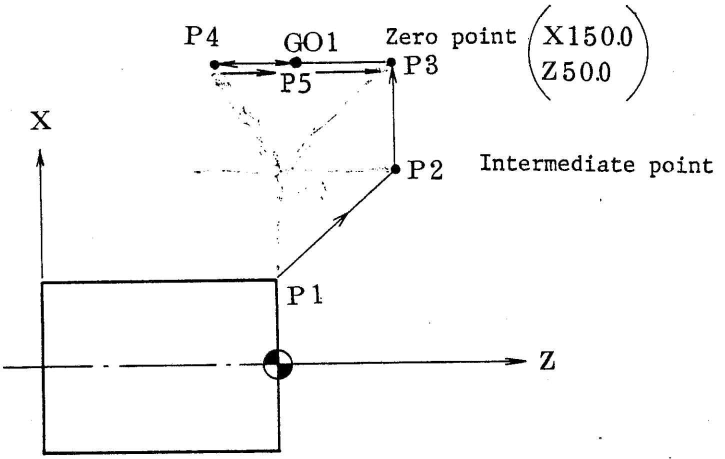
Example 1: Return to zero point through intermediate point.



Example 2: Return to zero point directly from starting point.



Example 3: Only commanded axis will travel to the intermediate point.



(X50.0 Z2.0;) P1

G28 X100.0 Z50.0; .... to P3 through P2

G01 Z-15.0; .... to P4

G28 ☐ Z2.0; .... to P3 through P5

When the ☐ part X-axis command is not given, only the Z-axis will travel in order of P4 → P5 → P3 since there is no intermediate point in the X-axis.

Note) Perform dog zero return for the axis which has not still been returned to zero point.

(3) G29 Automatic return from zero point

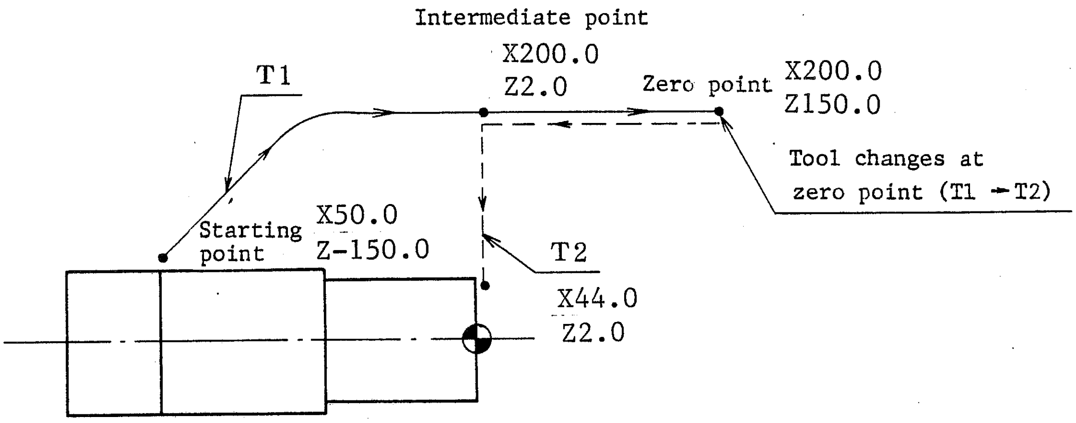
G29X(U) —— Z(W) ——;

This command provides positioning to the commanded position through an intermediate point, for commanded axes. In general, this command is used immediately following a G28 or G30 command.

At an incremental command, an incremental distance from intermediate point must be commanded.

With a G29 block, all commanded axes are moved to the intermediate point defined by the earlier G28 or G30 command at rapid traverse rate. The axes are moved to the commanded point at the rapid traverse rate.

Example of application of G28 and G29

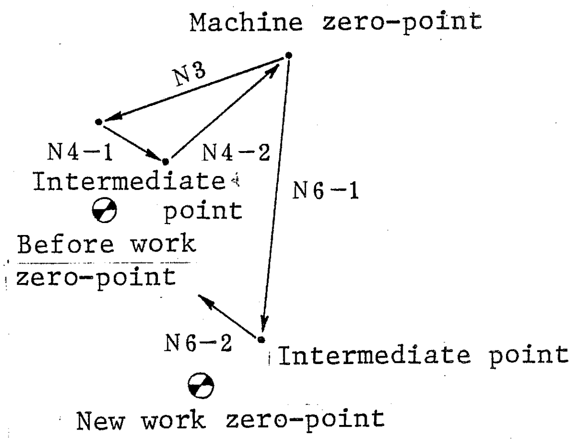


(X50.0 Z-150.0)	Starting point
G28 X200.0 Z2.0;	from starting point to zero point through commanded point (intermediate point)
T0202;	
G29 X44.0 Z2.0;	from zero point to commanded point through intermediate point

Note) If coordinate system is reset by the G50 command after a G28 or G30 command, the intermediate point will be relocated.

Examples)

```
N1 G28U0W0;
N2 G50X200.Z200;
N3 G00X100.Z0;
N4 G28X50.Z50.;
N5 G50X500.Z200.;
N6 G29X100.Z0;
    |
    |
    |
```



The intermediate point is a point in the coordinate system being used.

(4) G30 Fixed point return

The command below moves the commanded axes automatically to the fixed point.

$\boxed{\text{G30}}$X(U) —— Z(W) —— ;

The fixed point is determined by parameter setting as a distance from the machine zero point.

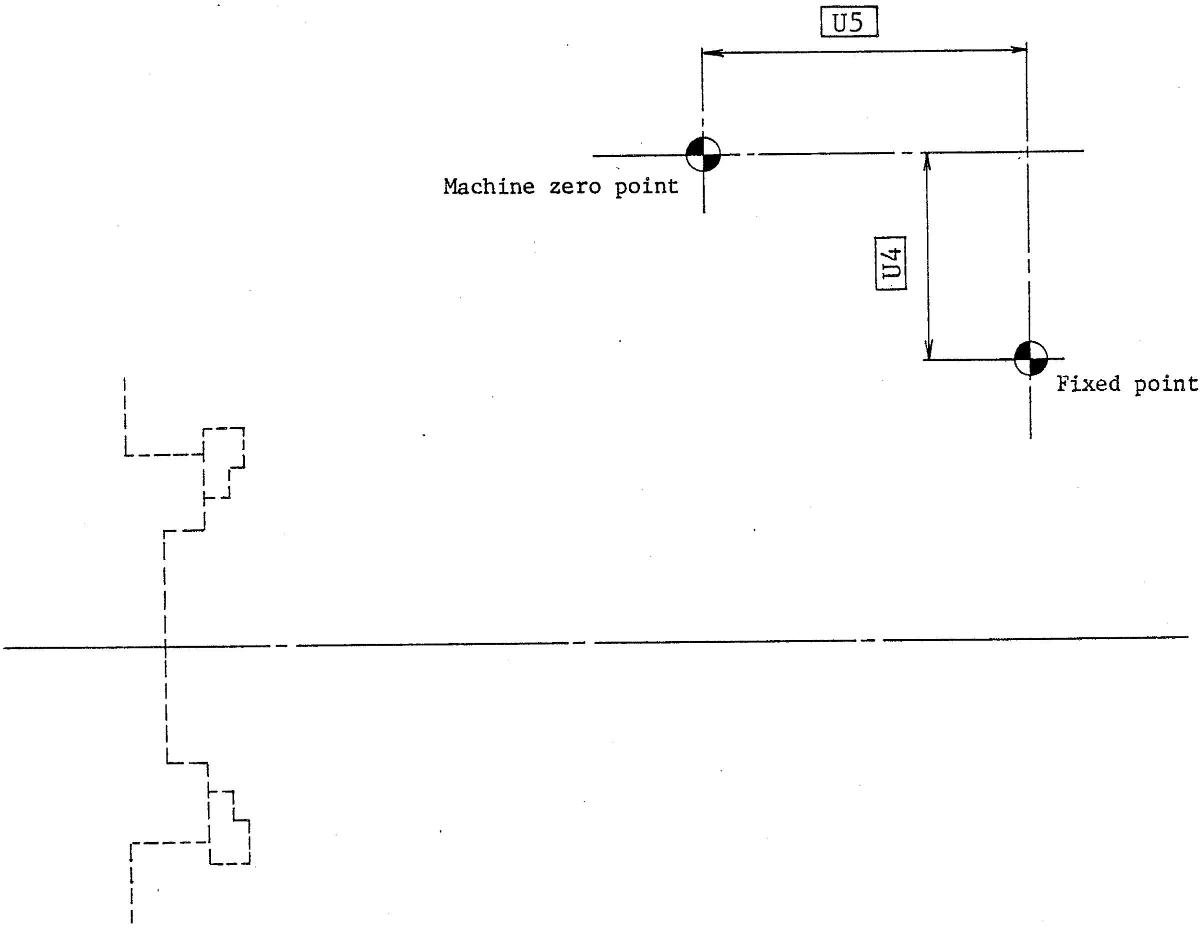
This function is the same as zero point return $\boxed{\text{G28}}$ except that a tool does not return to the machine zero point but to a fixed point. A $\boxed{\text{G29}}$ command following a $\boxed{\text{G30}}$ command positions a tool to the commanded position through the intermediate point which was established by the $\boxed{\text{G30}}$ command.

This motion is the same as that by a $\boxed{\text{G29}}$ following a $\boxed{\text{G28}}$ command. A $\boxed{\text{G30}}$ command is generally used when the automatic tool change (ATC) position differs from the fixed point.

Note) At least one manual or $\boxed{\text{G28}}$ zero point return is needed before giving a $\boxed{\text{G30}}$ command after turning on the power.

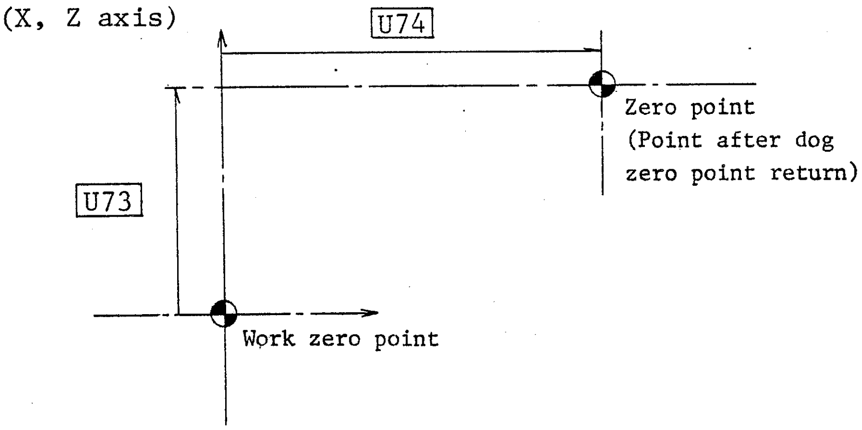
(5) Fixed point return position

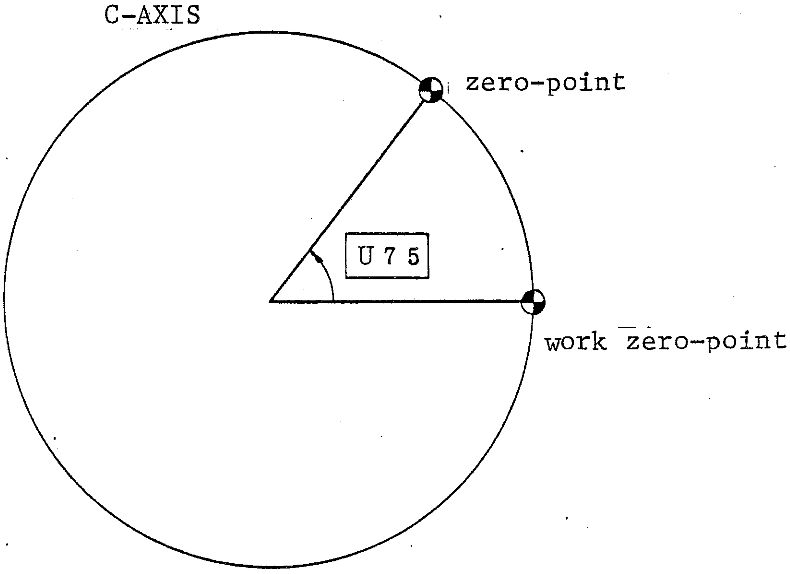
The fixed point position is set by parameters $\boxed{U4}$ and $\boxed{U5}$ as illustrated below.



(6) Work coordinate values automatically set when returning to zero point

Values of parameters $\boxed{U73}$ and $\boxed{U74}$ are used as work coordinate values of the zero point with respect to the work zero point on the completion of dog-type zero point return during automatic operation using EIA programs.





Example:

$\boxed{U73}$ = 100

$\boxed{U74}$ = 200

When the above values are set, X100. and Z200. are displayed at POSITION (current position counter) after dog-type zero point return.

7.9 Dwell (G04)

This command stops the machine operation during the commanded time.

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

Any one of these command is used for dwell. Upon completion of the previous block, (t) msec time elapses before beginning the next block.

Maximum commanded time is 9999.999 seconds. Error of time (t) is less than 16 msec.

Example: A dwell for 2.5 seconds
G04X2.5; or G04U2.5; or G04P2500;

Note 1) Address P cannot use decimal point programming.

Note 2) Dwell starts after the tool has arrived at the command value.

7.10 Mirror Image for Double Turrets (G68, G69)

A G code applies mirror image to the X axis.

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

Programming is made as follows for double turrets.

G50U —— (Distance between turrets);

Subsequent machining is done on the opposed turret. The program is, however, written as being machined on the turret of this side.

The machine is in G69 (mirror image cancel) status when the power is turned on.

7.11 Switching of Feedrate Command (G98, G99)

The following G codes specify either feed per minute or feed per revolution.

G code	Unit of feedrate
G98	Feed per minute
G99	Feed per revolution

The status when the power is turned on can be determined by parameter $\boxed{P9}$.

7.12 Constant Surface Speed Control (G96, G97)

The following G codes specify whether constant surface speed control effective or ineffective.

G code	CSSC	Meaning
G96	ON	Constant surface speed control is performed until G97 is commanded.
G97	OFF	Constant surface speed control is not performed until G96 is commanded.

CSSC : Constant surface speed control

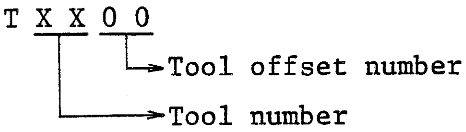
Note 1) When turning power on, the G97 (constant surface speed control off) is set.

8. COMPENSATION FUNCTION

When the tool different from the imagined tool is used in programming or an error corresponding to the tool nose roundness is corrected, the compensation function is used. This function contains tool offset and tool nose radius compensation.

8.1 Tool Offset

In T-2, tool offsets can be controlled by tool offset number included in T code. (Tool offset number: 1 - 64)
Tool selection and tool offset selection by T code:
T code has the following meaning:

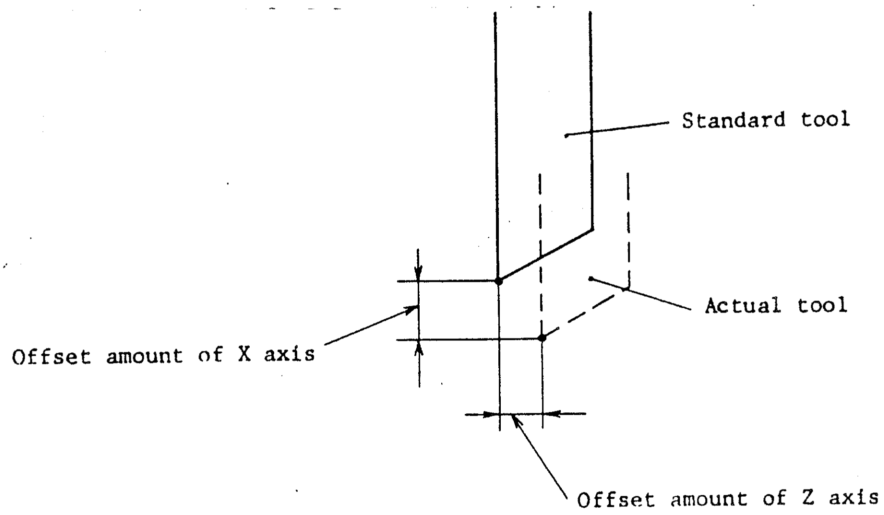


Example: T0203;

When above is run, tool No. 2 and tool offset No. 3 are selected.

8.2 Basic Tool Offset

The tool offset is used to correct the difference if any between programmed tool (usually standard tool) and the tool to be used actually.



8.3 Tool Offset Number

The tool offset number has two meanings:

The offset distance corresponding to the number is selected and the offset is started. The tool offset number 0 or 00 means that the offset amount is 0 and the offset is cancelled.

There are three types of compensation that correspond to the specified offset number.

Offset by the compensation of X, Z is called Tool Offset, and the compensation of R is called Tool Nose Radius Compensation.

Tool offset is effective when the T code is selected and its offset number is neither 0 nor 00.

Values that can be set as the offset distance are as follows:

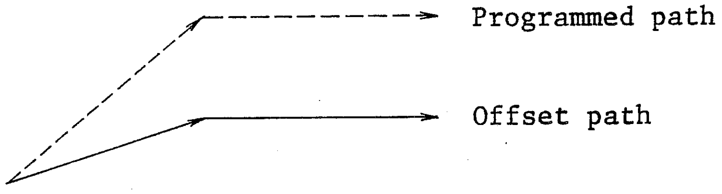
mm input	0	±9999.999 mm
inch input	0	±9999.999 inch

8.4 Offset

When only T-code is executed in one block, whether the tool moves by the offset distance in that block or it moves by that offset distance according to the next move command can be determined using parameter $\boxed{A6}$.

If the move command and T-code exist in one block, the tool moves to the position obtained by adding or subtracting the offset distance to or from programmed end point.

```
Example:  G00 X100. Z100. T0201; OFFSET-X -50. OFFSET-Z 0.
                Z200. ;
```



If 00 is specified for the offset number, the previous offset is cancelled. The end point of cancelled block is specified by the program.

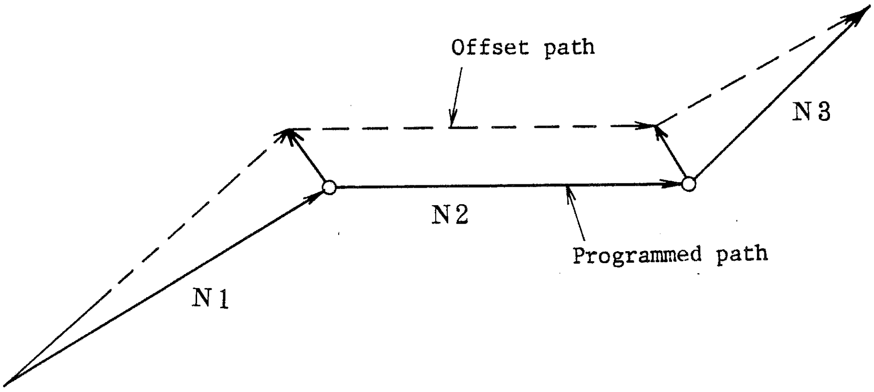
Note 1) If "G50X___. Z___. T___;" is specified, the coordinate system which comprises the coordinate value of (X, Z) is set. When the next move command is given, the tool moves by adding the offset distance for the offset number specified by T-code.

Note 2) If the offset distance is changed during the automatic operation, it becomes valid at the next block and on.

8.5 Offset Cancel

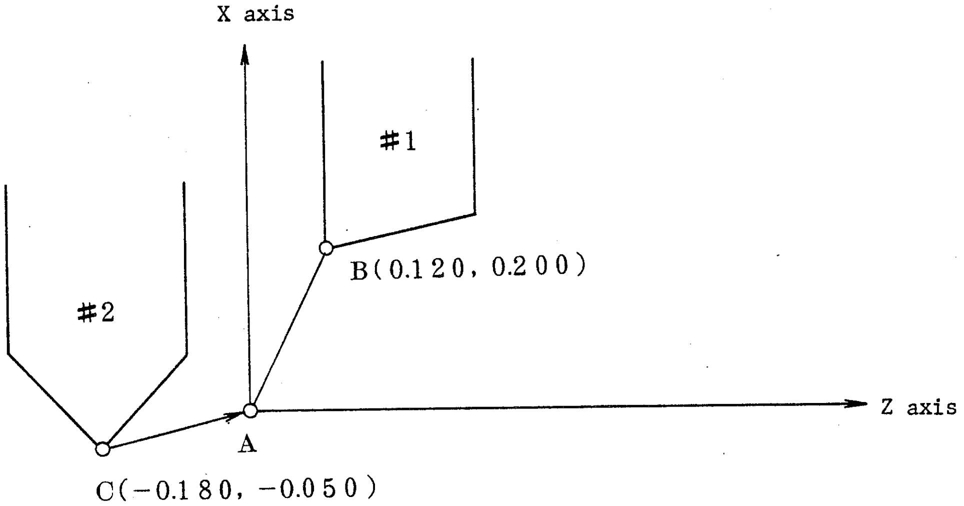
Offset is cancelled when the T code offset number 0 or 00 is selected. At the end point of the cancelled block, the offset vector becomes zero.

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



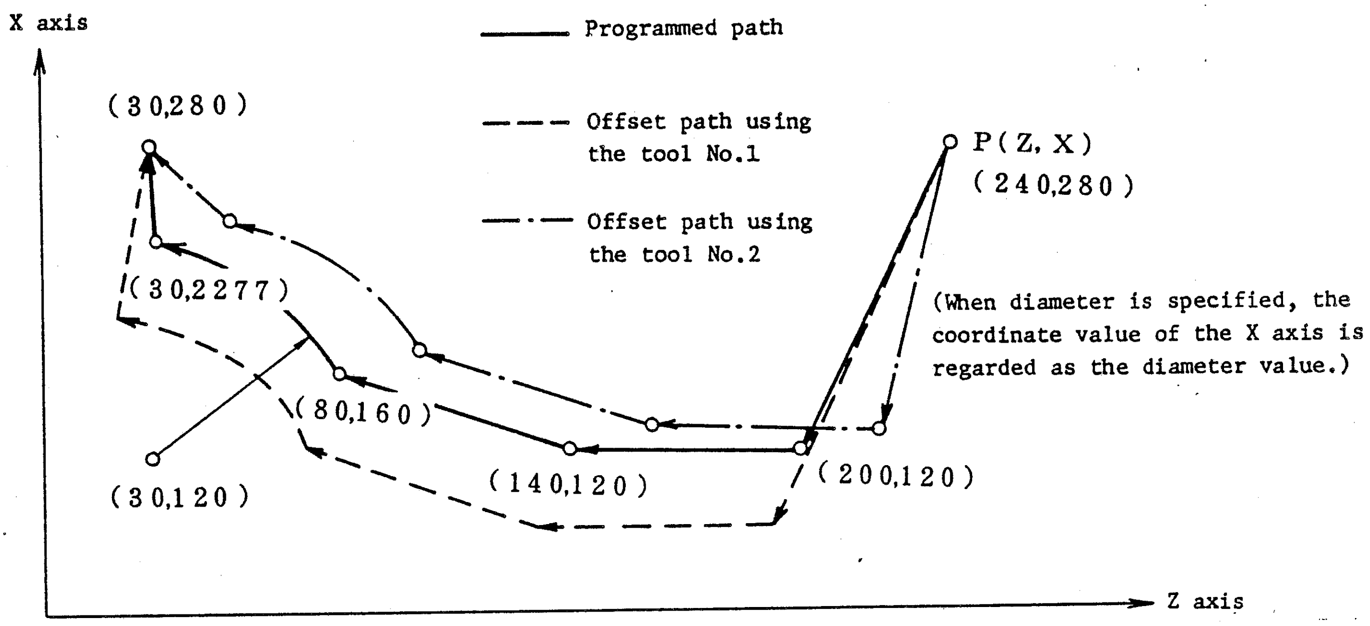
8.6 Program Example

	Tool nose coordinate value (Z, X)	Tool number
Tool No. 1	B (0.120, 0.200)	01
Tool No. 2	C (-0.180, -0.050)	02



A: Programmed tool nose
B: Actual tool nose (No.1)
C: Actual tool nose (No.2)

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



(When diameter is specified, the coordinate value of the X axis is regarded as the diameter value.)

(Program example 1)

```
G50 X 280.0 Z 240.0;
G00 X 120.0 Z 200.0 T0101;
G01 Z 140.0 F 30;
    X 160.0 Z 80.0;
G03 X 227.7 Z 30.0 I-20.0 K-50.0;
G00 X 280.0 T0100;
```

The path of the tool nose of tool No.1 coincides with the programmed path by this program.

(Program example 2)

The path of the tool nose of tool No.2 will coincides with the programmed path by the making following changes in program example 1.

T0101 → T0202
T0100 → T0200

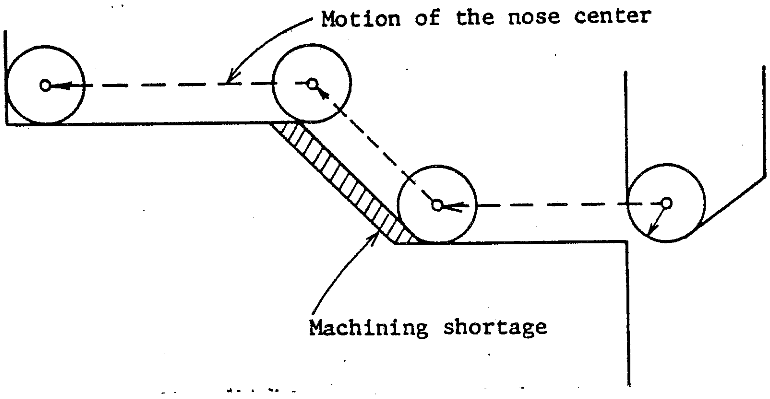
8.7 Tool Nose Radius Compensation (G40, G41, G42)

Tool nose radius compensations are made automatically for the machining shortage or excess corresponding to the roundness of the tool nose in taper or arc cutting.

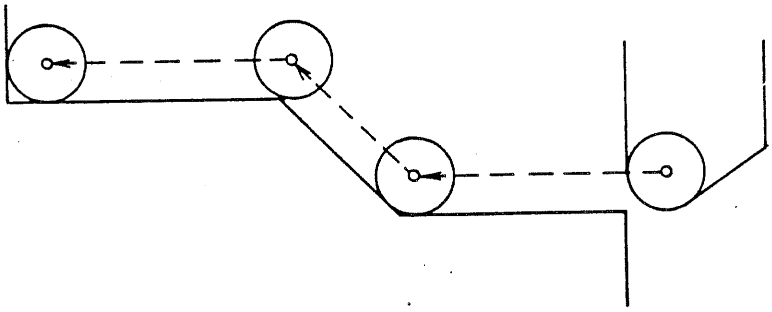
The following G codes are used:

G40: cancels tool nose radius compensation.
G41: runs tool nose to the right.
G42: runs tool nose to the left.

No tool nose radius compensation



Tool nose radius compensation

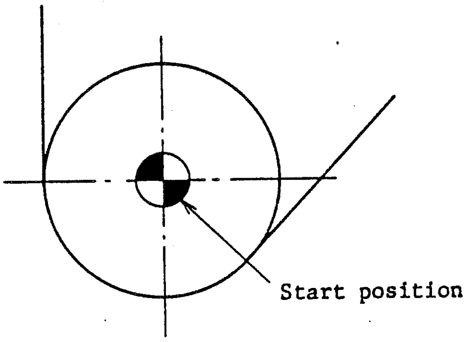


8.8 Tool Offset

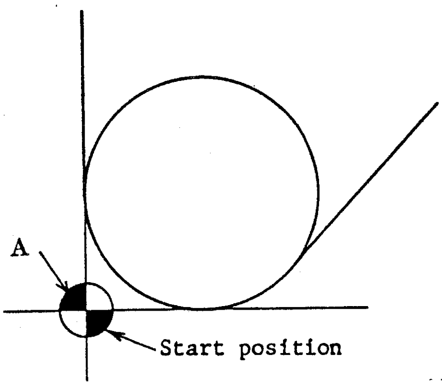
(1) Imaginary tool nose and tool nose center

The tool nose at position A in the following figure does not actually exist. The imaginary tool nose is required because it is usually more difficult to set the actual tool nose center to the start position or standard position than the imaginary tool nose. A tool without tool nose radius can be considered by the use of the imaginary tool nose.

The position relationship when the tool is set to the start position is shown in the following figure.



When programmed by the tool nose center



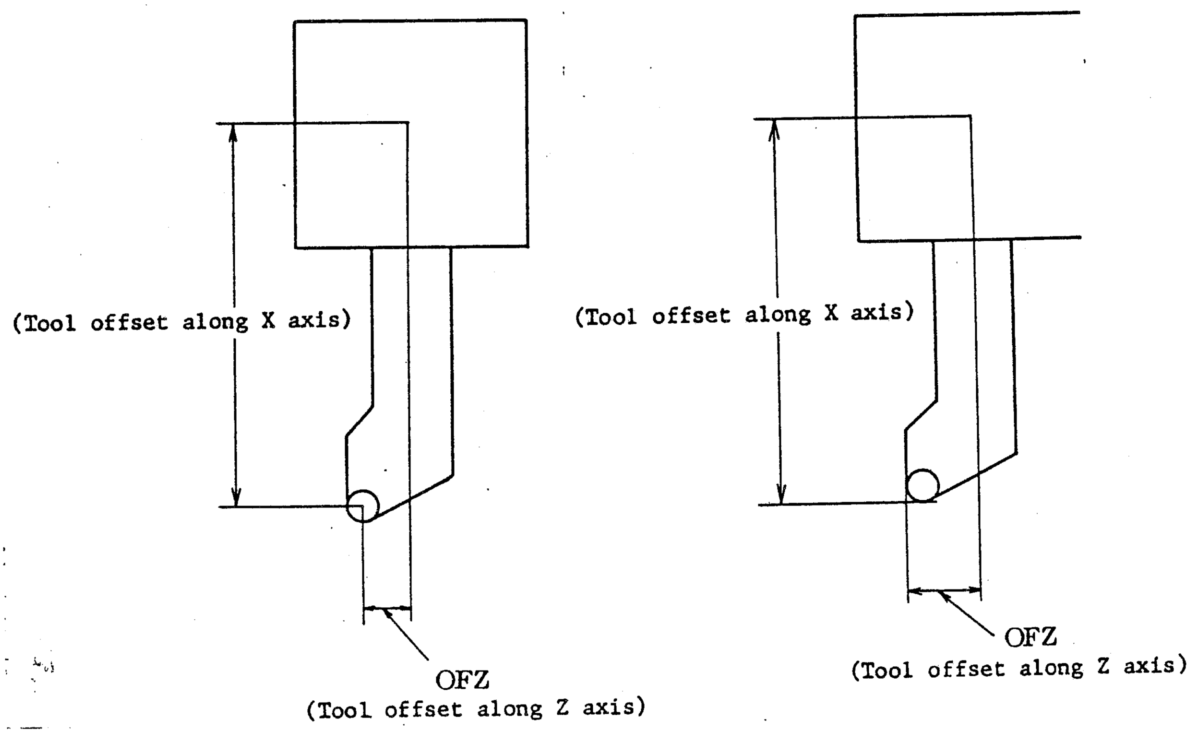
When programmed by the imaginary tool nose

The tool nose center can be placed at the start position with respect to a standard point such as the turret center. The distance from this standard point to the nose center or the imaginary tool nose is set as the tool offset value.

Setting the distance from the standard point to the tool nose center as the offset value is the same as placing the tool nose center at the start position. While setting the distance from the standard point to the imaginary tool nose is the same as placing the imaginary tool nose at the start position.

Usually, the distance from the standard point to the imaginary tool nose is set as the tool offset value.

- When placed at the start position with respect to the turret center.

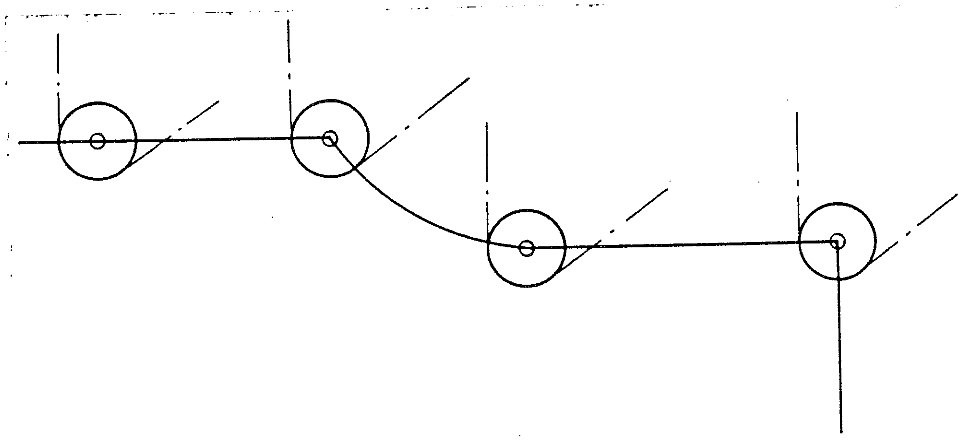


Setting the distance from the standard point to the tool nose center as the tool offset value is the same as placing the start position at the tool nose center.

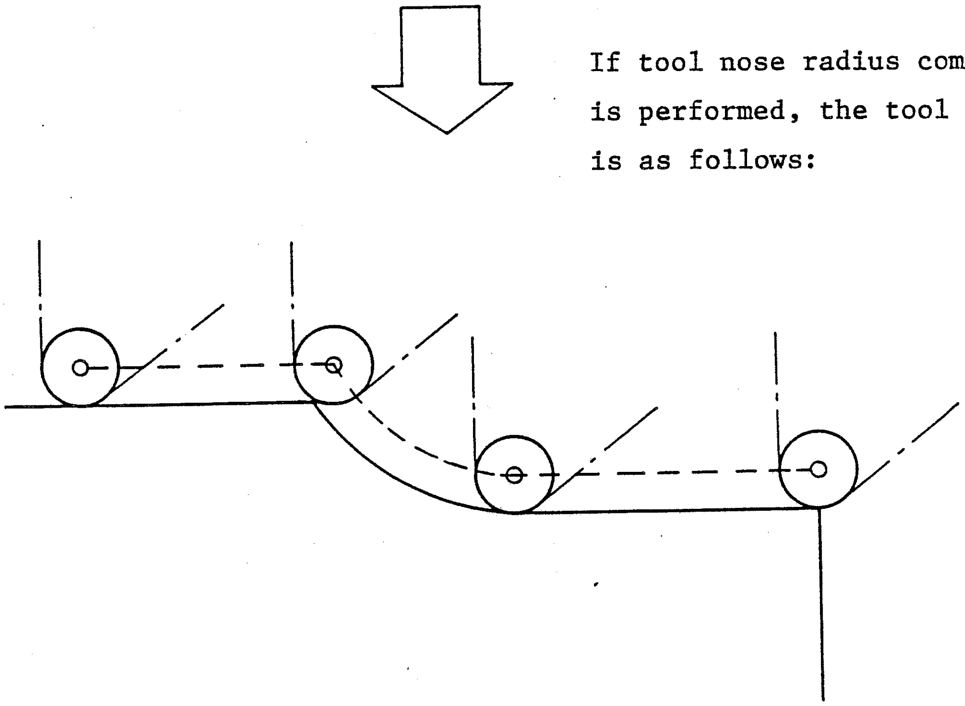
Setting the distance from the standard point to the imaginary tool nose as the tool offset value is the same as placing the start position at the imaginary tool nose.

(2) Programming by the tool nose center

Unless tool nose radius compensation is performed, the tool nose center path becomes the same as the programmed path. Tool nose path, when setting the tool offset value with the tool nose center placed at the start position:



If tool nose radius compensation is performed, the tool nose path is as follows:



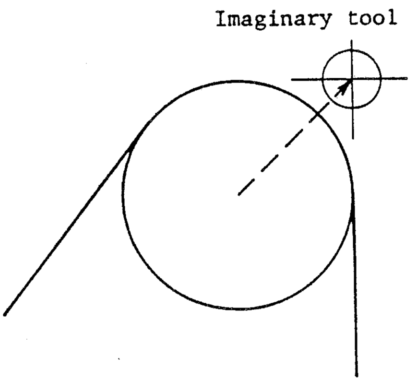
(3) Programming by imaginary tool nose

Unless tool nose radius compensation is used, the imaginary tool nose path becomes the same as the programmed path.

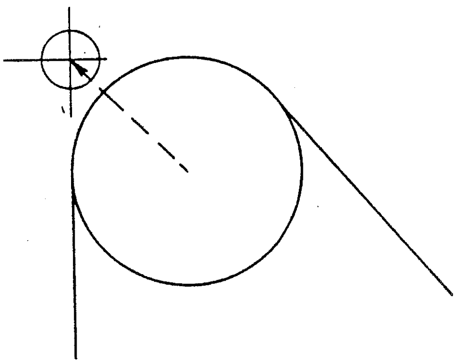
8.9 Setting of Imaginary Tool Nose Direction

The direction of the imaginary tool nose is set at "DIRCTN" in the TOOL OFFSET DATA display.

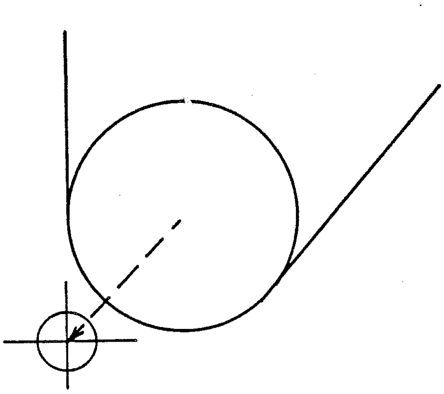
In the figure below, the relation between the tool and the start position is illustrated. The end of the arrow is the imaginary tool nose.



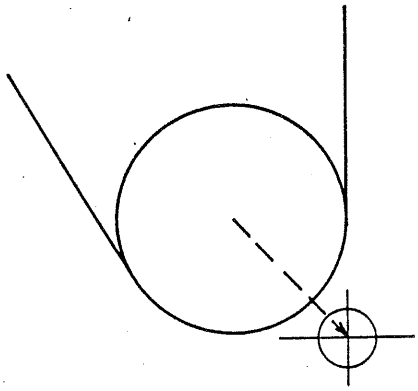
Imaginary tool nose direction number 1



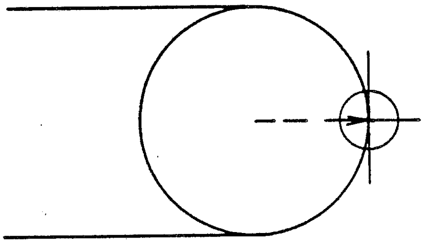
Imaginary tool nose direction number 2



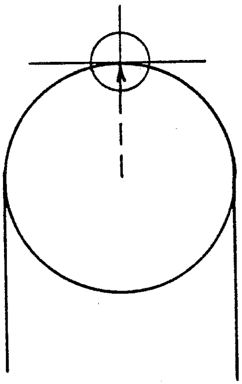
Imaginary tool nose
direction number 3



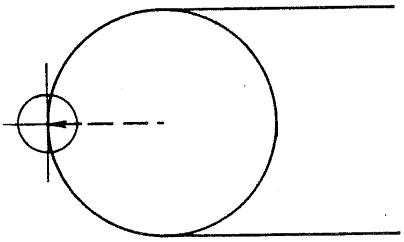
Imaginary tool nose
direction number 4



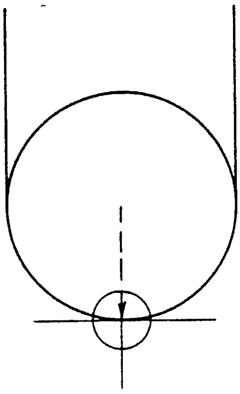
Imaginary tool nose
direction number 5



Imaginary tool nose
direction number 6

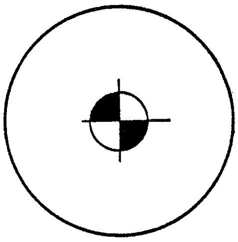


Imaginary tool nose
direction number 7



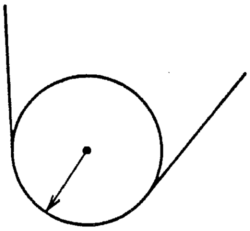
Imaginary tool nose
direction number 8

Imaginary tool nose direction numbers 0 and 9 are used when programming is made under the condition in which the tool nose center coincides with the start position.



Imaginary tool nose direction number 0 or 9

8.10 Setting of Tool Nose Radius Compensation Value (NOSE-R)



Tool nose radius compensation value (Tool nose radius value)

The range specified as the offset value is as follows:

	Input in metric system	Input in inch system
Offset value	0 - ±9999.999 mm	0 - ±9999.999 inch

The offset value corresponding to the offset number 00 in the T code is always zero.

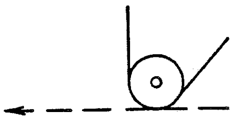
8.11 Direction Command in Tool Nose Radius Compensation

In tool nose radius compensation, the position of the work with respect to the programmed path must be commanded.

G code	Work position	Tool path
G40	(Cancel)	Moving on the programmed path
G41	Right side	Moving at the left side of the programmed path
G42	Left side	Moving at the right side of the programmed path

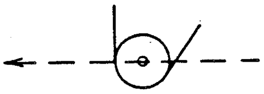
G40

(Imaginary tool nose direction number 1 - 8)



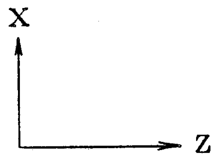
The imaginary tool nose is on the programmed path.

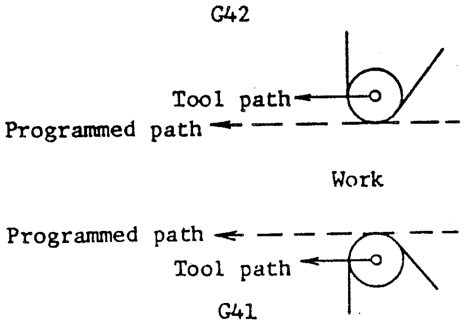
(Imaginary tool nose direction number 0 or 9)



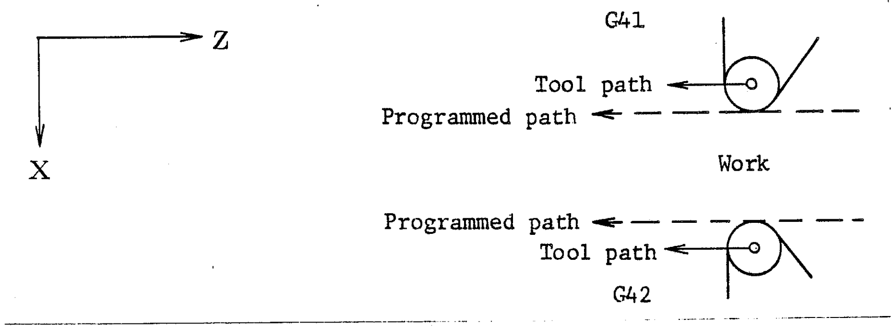
The tool nose center is on the programmed path.

G41, G42





If the coordinate system setting is changed as shown below or the offset value is negative, the work position is changed as follows:



G40, G41 and G42 are modal. When the power is turned on or the (RESET) key is pressed, G40 is effective.

8.12 Precautions when Compensating the Tool Nose Radius

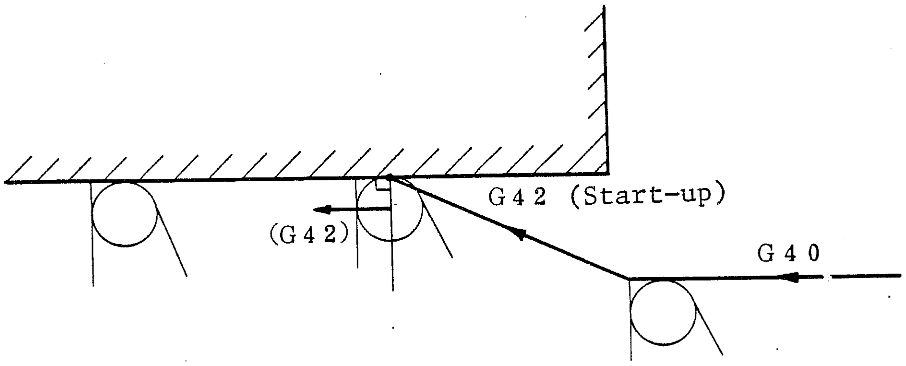
(1) Tool nose approach in start-up

The block in which the mode is first changed to G41 or G42 from G40 (tool nose radius compensation cancel) mode is called start-up block.

```
G00 G40 X50.0 Z3.0;
G01 G42 X60.0 Z-20.0; (Start-up block)
        Z-40.0;
```

In the start-up block, the transient tool movements for offset are performed.

In the block after the start-up block, the tool nose center at the starting point is positioned vertically to the programmed path of that block.

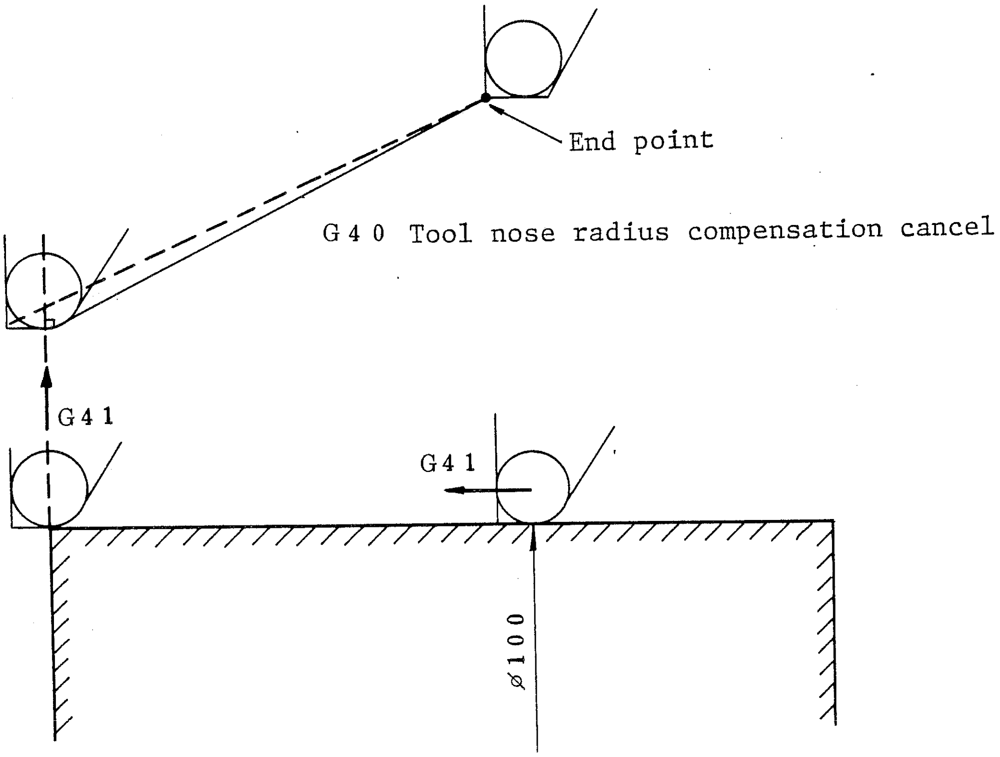


8.13 Tool Nose Radius Compensation Cancel

The block in which the mode is changed to G40 from G41 or G42 mode is called tool nose radius compensation cancel block.

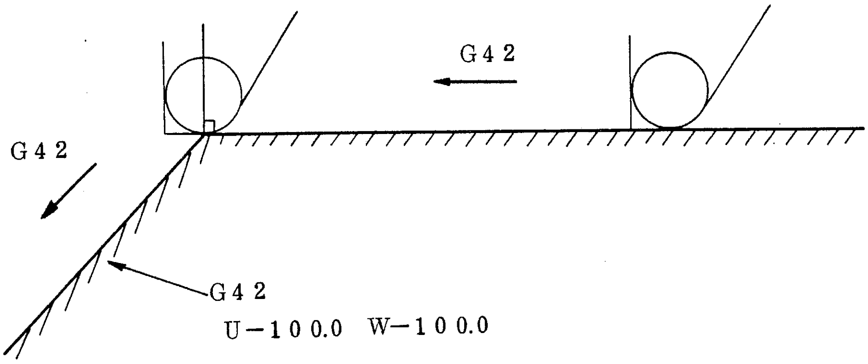
```
G42 G01 Z-50.0;
        X100.0 Z-70.0;
        X120.0;
G40 G00 X200.0 Z-50.0; (Tool nose radius compensation cancel)
```

The tool nose center at the end point comes to the position vertical to the programmed path in the block before the cancel block. The imaginary tool nose is positioned at the end point in the cancel block (G40) as shown below.



8.14 When G41/G42 is Again Commanded in G41/G42 Mode

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



At the block in which the G41/42 is first commanded (in start-up), the above positioning of the tool nose center is not performed.

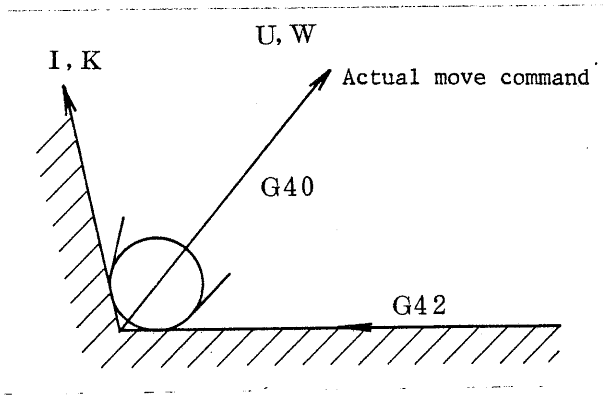
8.15 When the Moving Direction of the Tool in the Block which Includes a G40 Command (Tool Nose Radius Compensation Cancel) is Different from the Direction of the Work Shape

When the tool is to be retracted during machining in the direction to point (A) with cancelling the tool nose radius compensation in the figure below, command as follows:

G40X —— Z —— I —— K —— ;
or G40U —— W —— I —— K —— ;

In the above command, I and K are specified by the incremental programming in the direction corresponding to the work shape in the next block.

Note) Unless I and K have been specified, the tool bites the work in the tool nose radius compensation cancel mode.



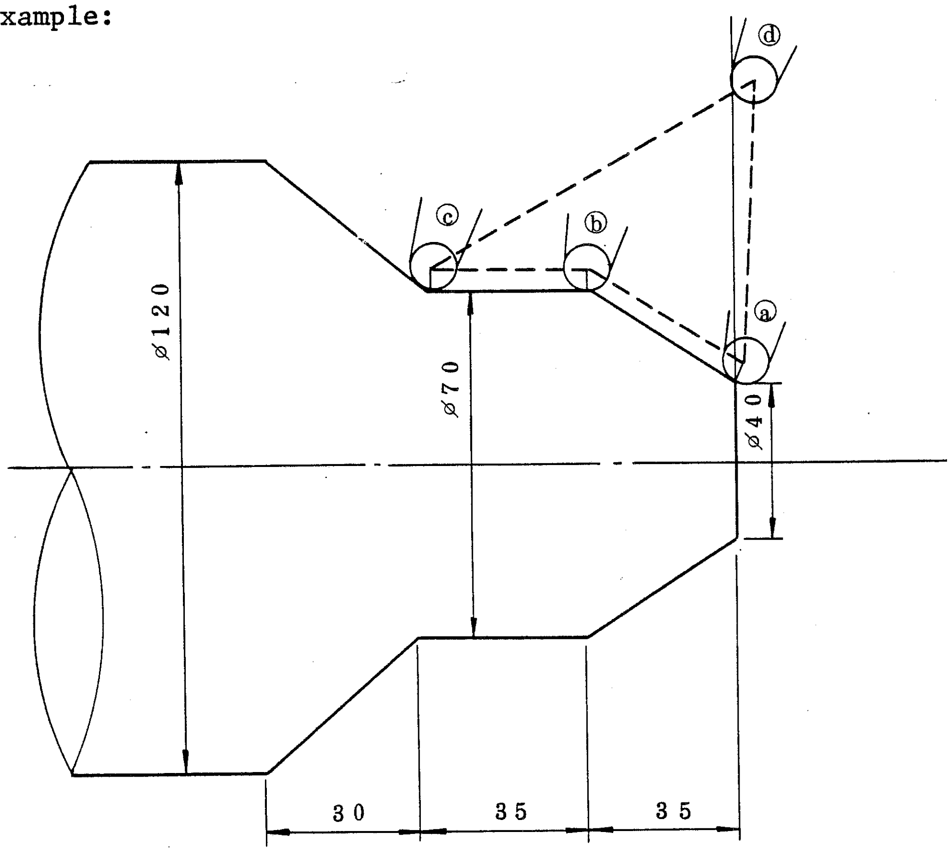
G40U —— W —— I —— K ;

The work position specified by the addresses I and K is the same direction as that in the G41 or G42 mode.
If I and K are specified with the G40 in the cancel mode, the I and K are ignored. The following is a command in the tool nose radius compensation cancel mode:

G40 G01 X —— Z —— ;	Same as G01 X —— Z —— ;
G40 G00 X —— Z —— I —— K —— ;	Same as G00 X —— Z —— ; I —— K —— : ineffective
G00 X —— Z —— I —— K —— ;	AS G40 is not commanded, I and K are regarded as chamfering data. But, as this format is erroneous, an alarm is displayed.

The numerals under I and K should always be specified by radius values.

Example:



(G40 mode)

```
(a)  G42 G00 X40.0;
(b)  G01 X70.0 Z-35.0 F0.2;
(c)  Z-70.0;
(d)  G40 G00 X150.0 Z3.0 I25.0 K-30.0;
```

8.16 Two or more Blocks without a move Command

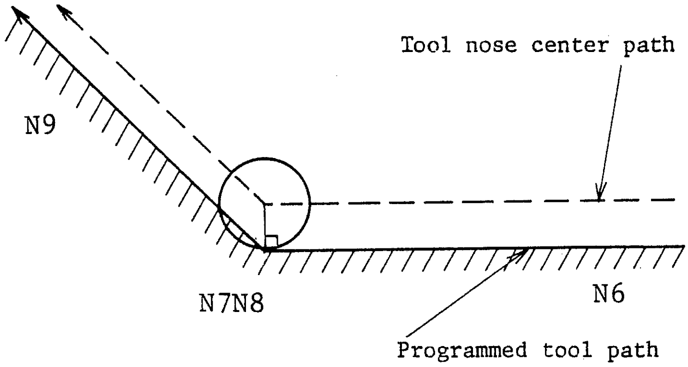
Blocks without a move command are as follows:

(a)	M00;	M code
(b)	G04 X3000;	Dwell
(c)	S120;	S code
(d)	G00 U0;	Feed distance is zero.
(e)	G99;	G code only

Two or more blocks with a offset distance change

```
G10 P01 X 1.0 Z0.5 R0.8 Q4;
     G10 P05 X-1.6 Z0.2 R0.4 Q3;
```

If two or more of the above blocks are commanded consecutively, the tool nose center comes to the position vertical to the programmed path of the preceding block at the end of the preceding block. However, if the no-movement command is (d) in the above, above tool motion is attained only by a block without move distance.



```
(G42 mode)

N3 Z-100.0;
N4 S150;
N5 G04 X2000;
N6 X112.0 Z-130.0;
```

As shown in above example, machining excess will occur.

Do not insert command blocks such as G10, M00/M01 while compensating the tool nose radius.

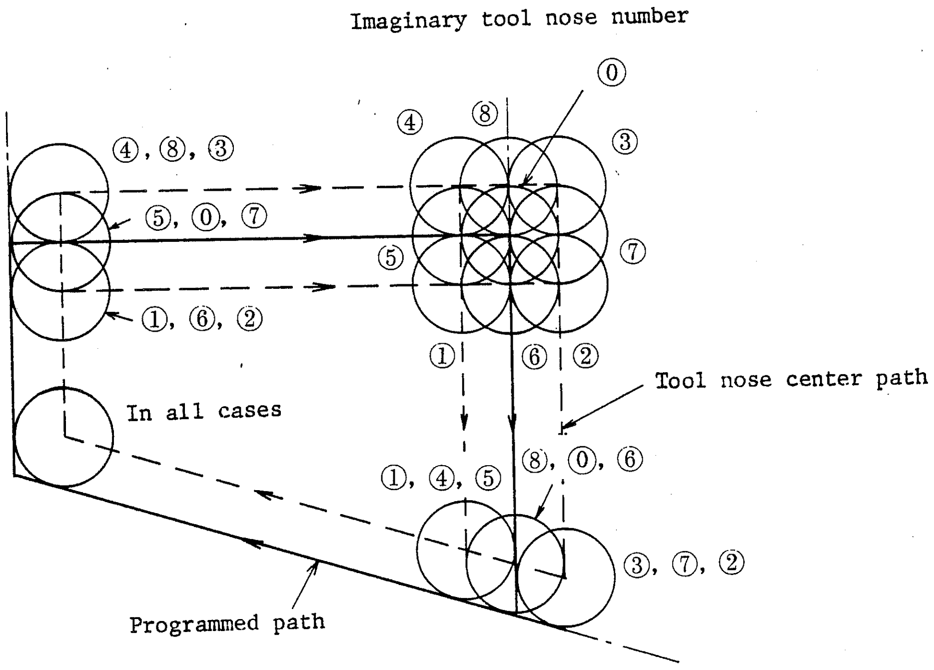
8.17 Compensation with G90 or G94

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

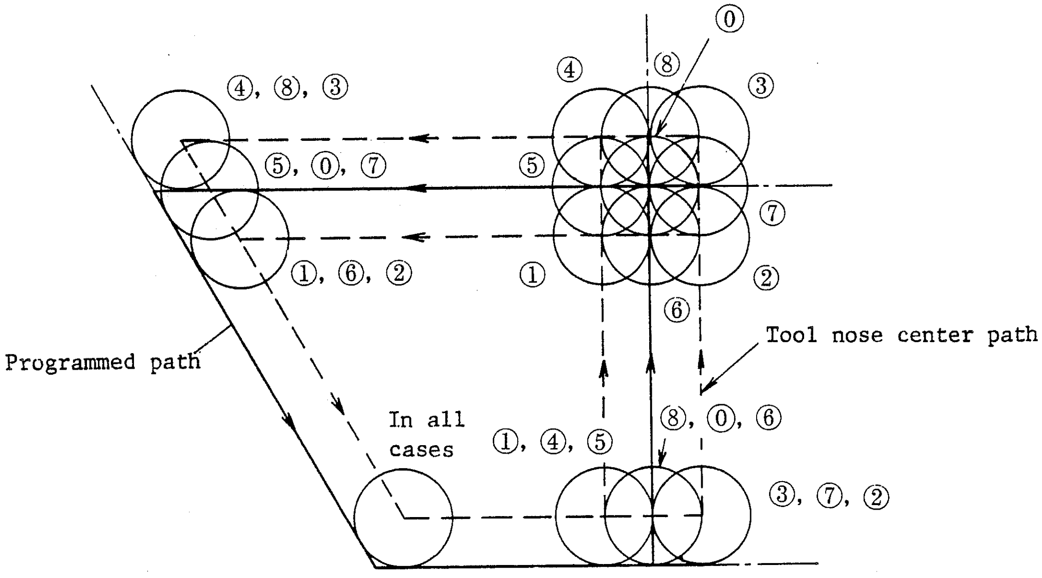
(a) Motion for each imaginary tool nose position

Tool nose center path is generally parallel to the programmed path.

(i) G90

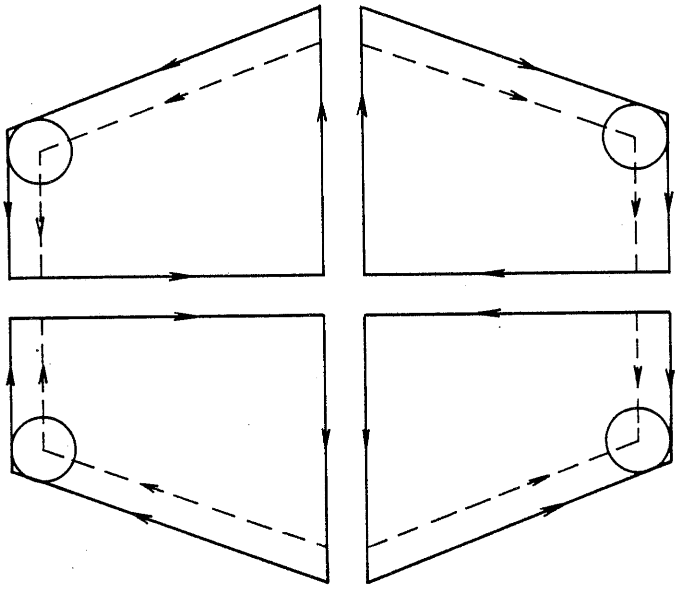


(ii) G94

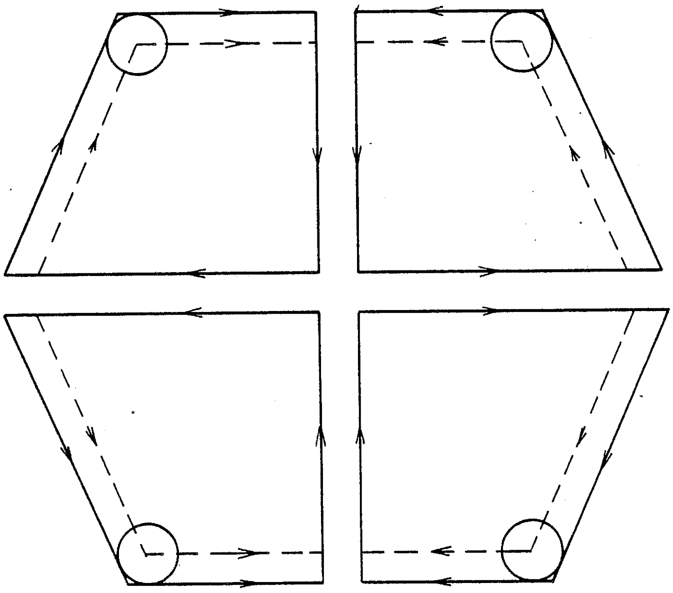


(b) The offset direction is indicated in the figure below, regardless of G41/G42 mode.

(i) G90



(ii) G94



8.18 Compensation with G73

When machining is made under the condition in which the tool nose center is regarded as imaginary tool nose, if the machining is performed by G73 with tool nose radius compensation, the tool nose radius compensation value is added to the finishing allowance U and W but the tool nose radius compensation is cancelled during rough machining.

That is,

New finishing allowance U'

= U + Tool nose radius compensation value (U = 0)

New finishing allowance W'

= W + Tool nose radius compensation value (W = 0)

Refer to Section 9.7.

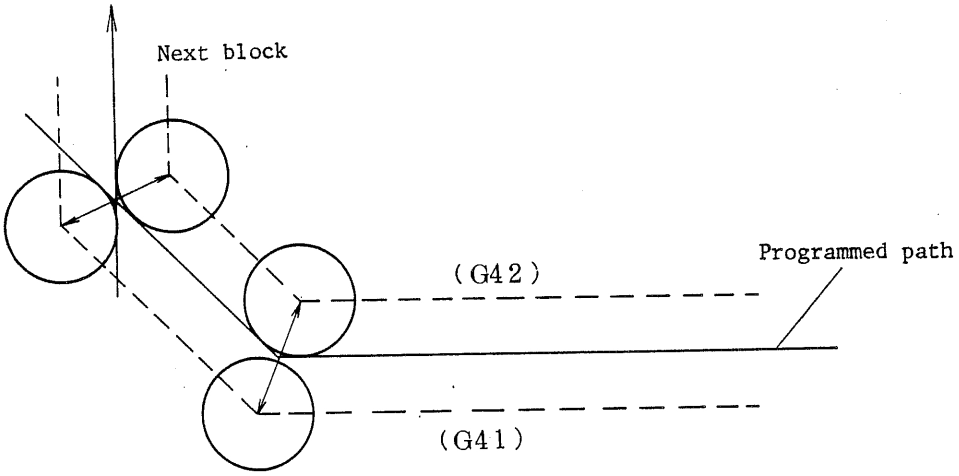
When finishing allowances U and W are zero or the imaginary tool nose coincides with the tool nose, the tool nose radius compensation value is not added to the finishing allowance. Other tool nose compensation than the above is ignored.

8.19 When G71, G72, G74 - G76 or G92 is Commanded.

Tool nose radius compensation is not performed in this case.

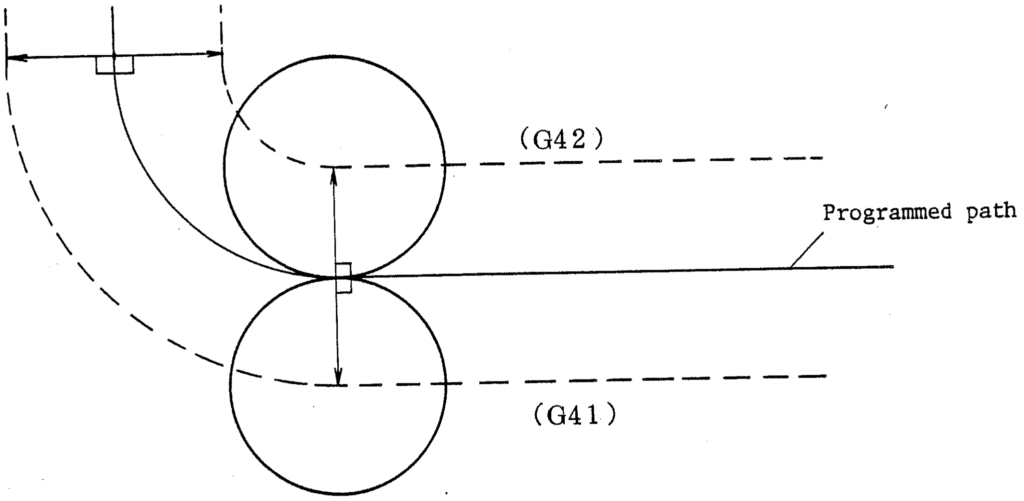
8.20 When Chamfering is Performed.

Movement after compensation is shown below:



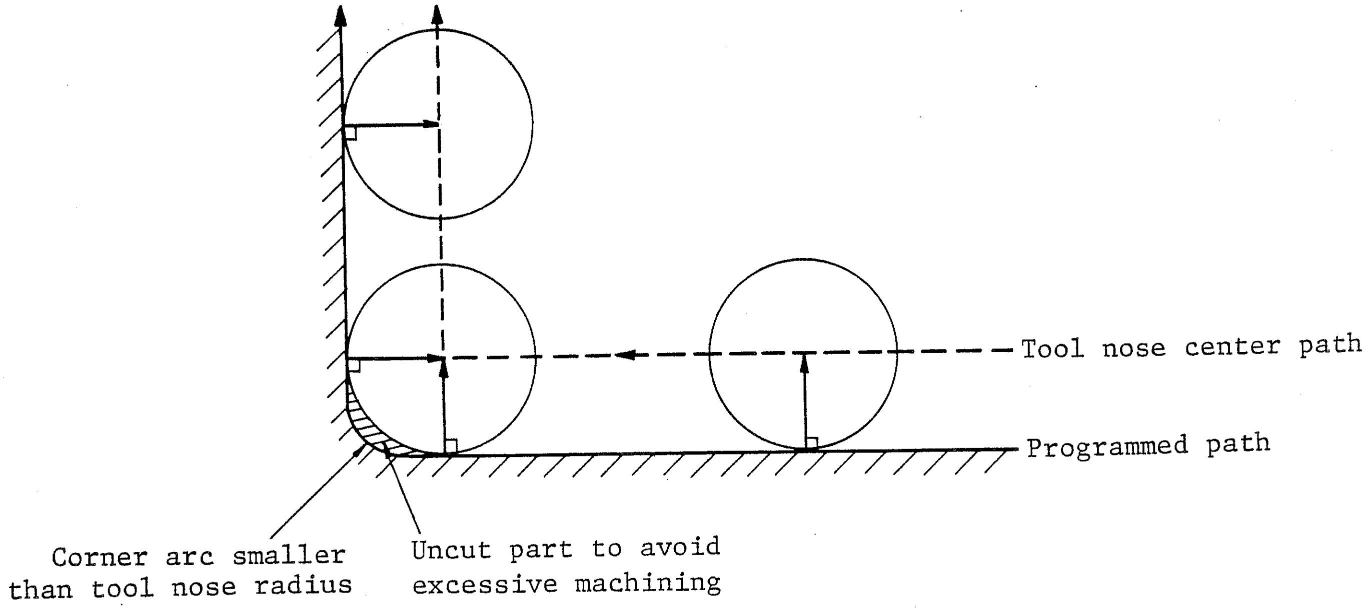
8.21 When a Corner Arc is Inserted

Movement after compensation is shown below:



8.22 When the Machining is Performed at an Inside Corner whose Arc Radius is Smaller than the Tool Nose Radius

In this case, the inner offsetting of the tool causes the tool nose center to draw the path in the figure below.
Uncut parts will remain but machining excess will not occure.

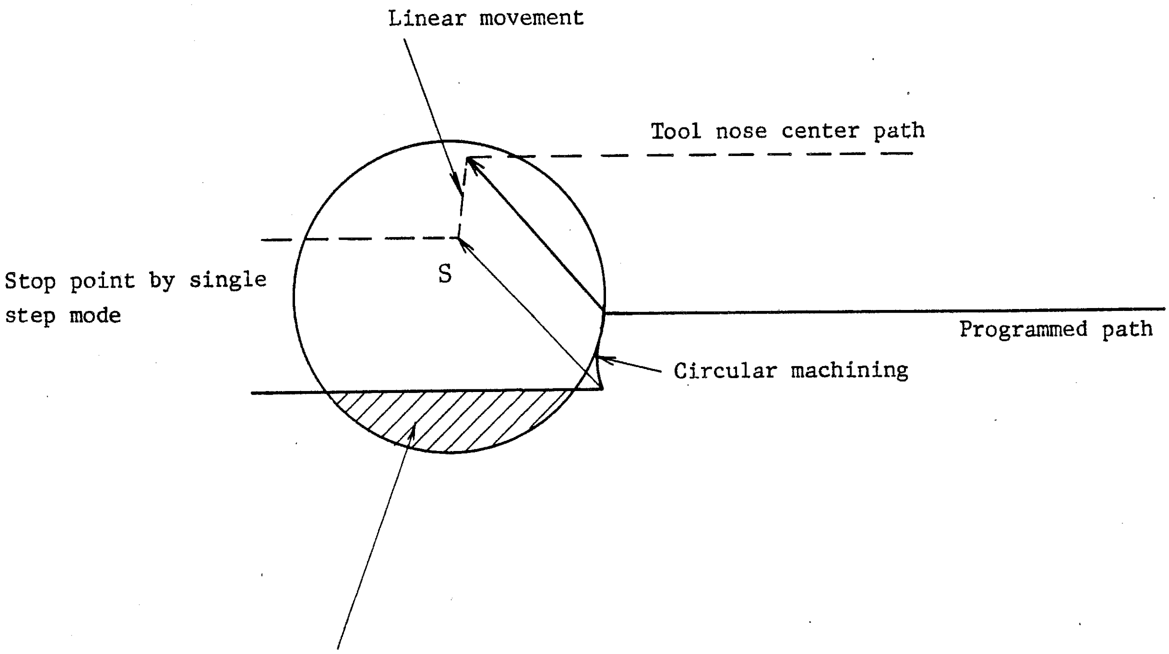


If a tool nose radius compensation value is larger even under a condition that machining excess will not occure, alarm 636 "NOSE R OFFSET ERROR" will be displayed.

8.23 When Machining a Step Smaller than the Tool Nose Radius

In the case of a program containing a step smaller than the tool nose radius and when the step is an arc, the path of the tool nose center may be reversed to the programmed direction. In this case, the first vector is ignored, and the tool moves linearly to the second vector position. The motion is stopped at this point in the single step mode. If the single step mode is not on, the automatic operation will continue.

If the step is specified with a line, the offset is properly done without alarm. (However, uncut parts remain.)



Because the first vector is ignored, this overcutting will not generate.

8.24 Program Input of Tool Offset Amount (G10)

Tool offset values can be input by program. The command is as follows:

G10P —— X —— Z —— R —— Q —— ;
or G10P —— U —— W —— R —— Q —— ;

P : Tool offset number
X : Offset value in X axis (absolute)
Z : Offset value in Z axis (absolute)
U : Offset value in X axis (incremental)

W : Offset value in Z axis (incremental)
R : Tool nose radius compensation value (absolute)
Q : Imaginary tool nose direction number

In absolute command, the values specified by addresses X and Z are set as an offset value corresponding to the offset number specified by address P.
In incremental command, the values specified by addresses U and W are added to the stored offset amount corresponding to the offset number specified by address P.

Note 1) Addresses X, Z, U and W are allowed to be commanded in the same block.

Note 2) Specifying this command (G10) in a machining program allows the tool to advance little by little such as in thread cutting. Apart from a machining program, this command can also be used to input offset values at a time, from a tape in which they are punched by specifying this command succesively, instead of inputting them one at a time using number keys.

Note 3) Even though tool offset program input is performed on the TOOL OFFSET DATA display, the inputted results can only be confirmed on that display after another display is selected.

9. MACHINING CYCLE FUNCTION

For repetitive machining peculiar to turning, such as metal removing in roughing, a series of paths that is specified usually in a range of dozens of blocks can be specified in one block.

The machining cycle is of the following two types.

(i) Canned cycle

(ii) Multiple repetitive cycle

Note) The drawings shown in the examples below are illustrated in diameter programming.

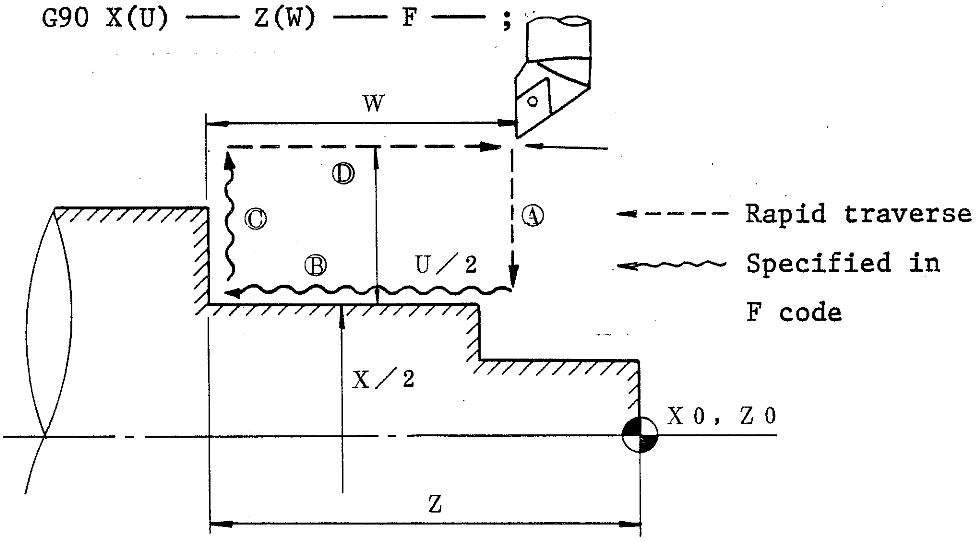
9.1 Canned Cycle (G90, G92, G94)

There are three canned cycles, G90, G92 and G94.

9.2 Cutting Cycle A, G90

(1) The straight cutting cycle is programmed by the following command:

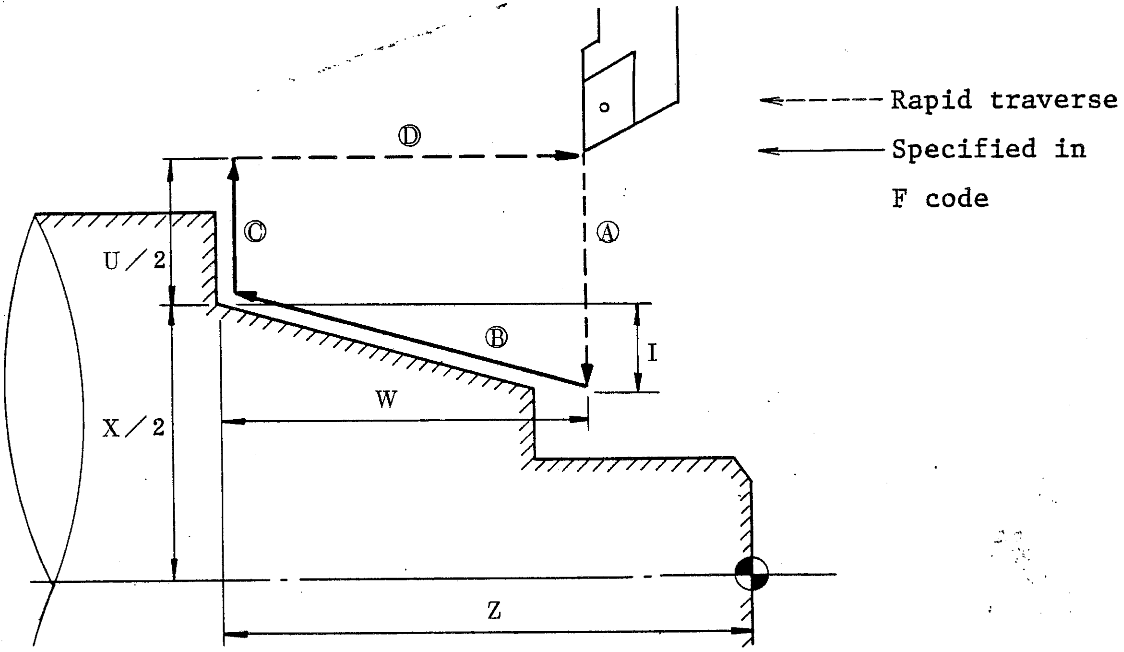
G90 X(U) —— Z(W) —— F —— ;



In single step mode, (A), (B), (C) and (D) are performed separately. In incremental programming, the signs of the numeric values following address U and W depend on the direction of pathes (A) and (B). In the above figure, those signs are negative.

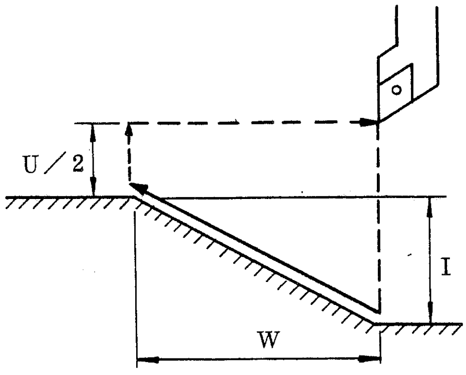
(2) The taper cutting cycle is programmed by the following command:

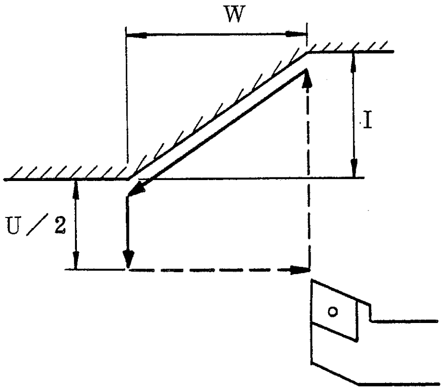
G90 X(U) —— Z(W) —— I —— F —— ;

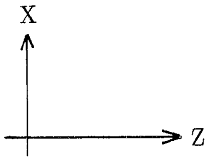


In incremental programming, the relationship between the signs of the numeric values following addresses U, W and I, and the tool paths is as follows:

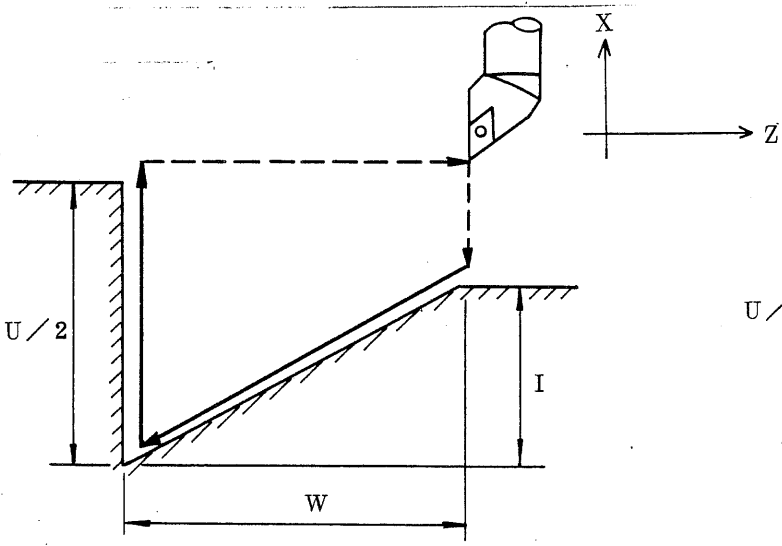
(i) $U < 0, W < 0, I < 0$ (ii) $U > 0, W < 0, I > 0$

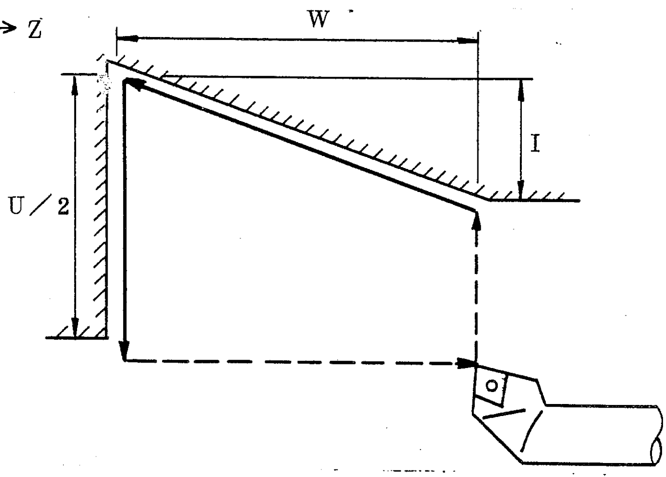






(iii) $U < 0,\ W < 0,\ I > 0$ at $|I| \leqq \left|\frac{U}{2}\right|$ (iv) $U > 0,\ W < 0,\ I < 0$ at $|I| \leqq \left|\frac{U}{2}\right|$

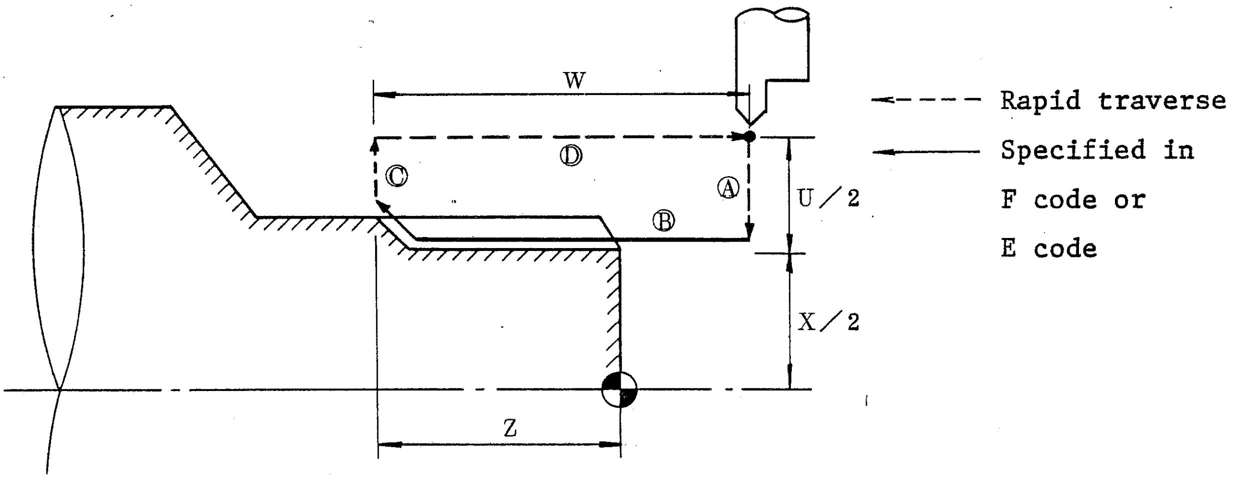




9.3 Thread Cutting Cycle (G92)

(1) Straight thread cutting is programmed by the following command:

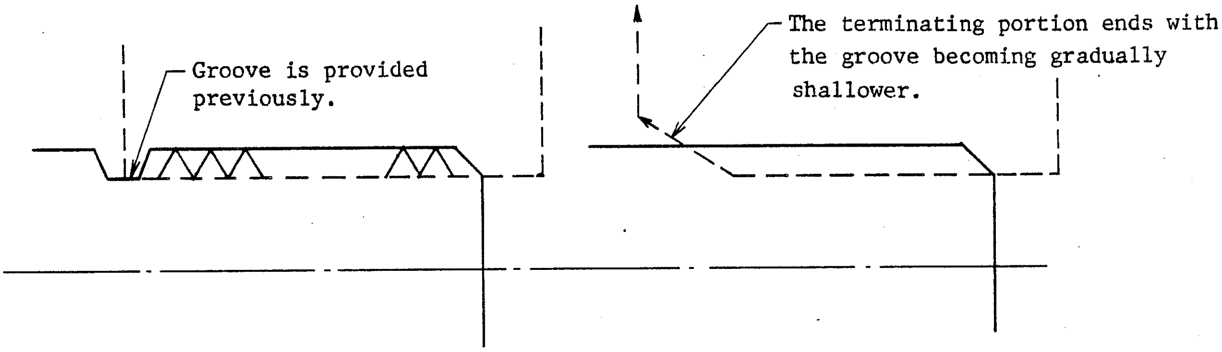
```
                         ┌ F ── ;
G92 X(U) ── Z(W) ──  ┤
                         └ E ── ;
                           └──Thread lead is specified.
```



In incremental programming, the signs of numeric values following addresses U and W depend on the direction of paths (A) and (B). That is, if the direction of path (A) is the negative direction of X axis, the value of U is negative.
The range of thread leads, limitation of spindle speed, etc. are the same as in G32 (thread cutting).
In this thread cutting cycle, thread chamfering is possible.

By the use of an M code, thread chamfering is performed. This chamfering distance r is set in the range from 0.1L to 4.0L in 0.1 increments by parameter U36. (In the above expression, L is the thread lead.)
In single step mode, (A), (B), (C) and (D) are performed separately.

(2) Application
The chamfering eliminates the necessity of avoiding an incomplete thread by machining a thinner groove than the thread root at the terminating portion of thread.
This function increases the strength of the screw, and is useful for a screw requiring sealing.



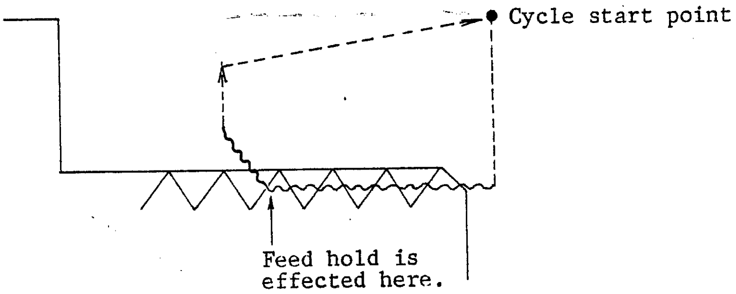
(a) When thread is not chamfered (b) When thread is chamfered

Note) Notes on this thread cutting are the same as in thread cutting in G32.
However, a stop by feed hold during thread cutting is as follows:

1) When "0" is set for parameter $\boxed{\text{U24}}$.
Feed hold ON ⟶ Stop after completion of operation in path (C)

2) When "1" is set for parameter $\boxed{U24}$.

As soon as feed hold is effected during threading, the tool carries out chamfering and stops at the position A1 where the threading is finished. Depressing the ☐ (CYCLE START)key will return the tool to the starting point by the simultaneous 2-axes rapid feed. Then, the tool performs threading again at the similar position.



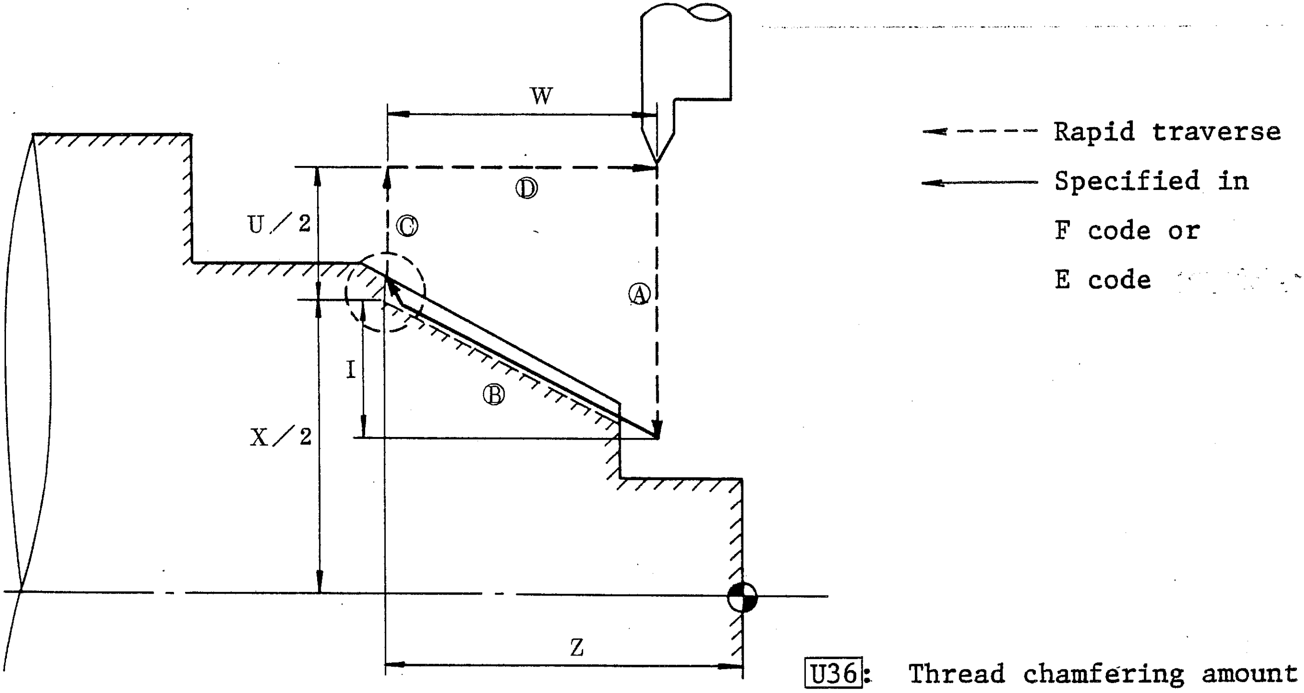
* Parameter $\boxed{U24}$ is effective for G92 and G76 only.

Another feed hold is ineffective during chamfering. The chamfered amount during retreat is the same as that at the end point.

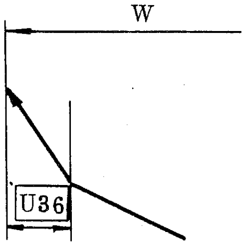
(3) Taper thread cutting is programmed by the following command:

$$\text{G92 X(U) —— Z(W) ——}\left\{\begin{array}{l}\text{F —— ;}\\ \text{E —— ;}\end{array}\right.$$

Thread lead is specified.



$\boxed{\text{U36}}$: Thread chamfering amount (Parameter)

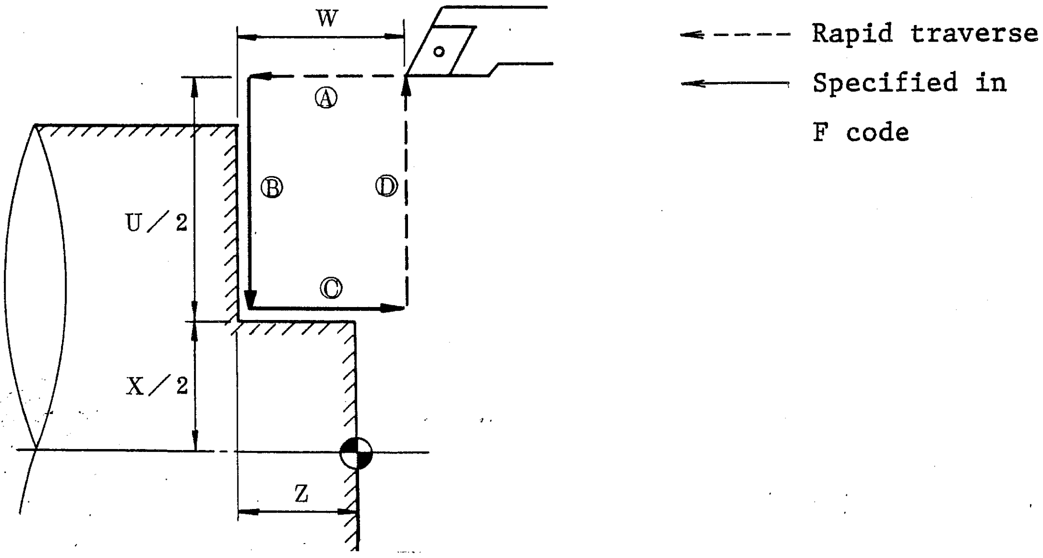


Detailed chamfered thread

9.4 Cutting Cycle B (G94)

(1) The face cutting cycle is programmed by the following command:

G94 X(U) —— Z(W) —— F —— ;

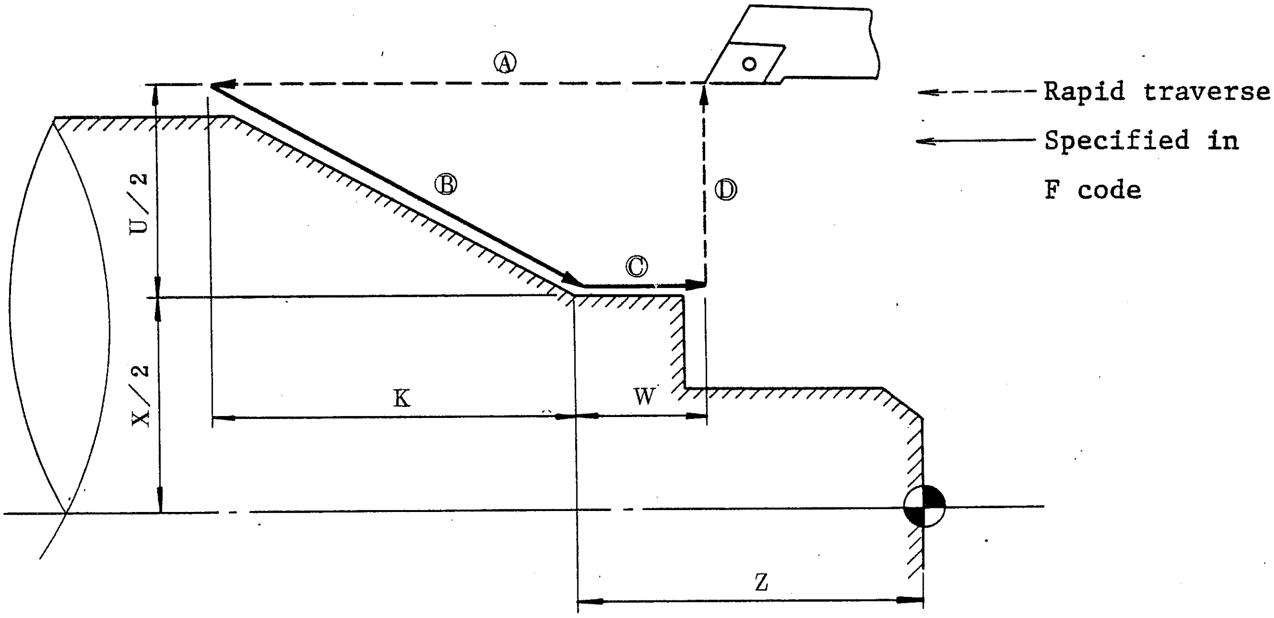


In incremental programming, the sign of numeric values following addresses U and W depends on the direction of the paths (A) and (B). That is, if the direction of the path (A) is in the negative direction of the Z axis, the value of W is negative.

In single step mode, (A), (B), (C) and (D) are performed separately.

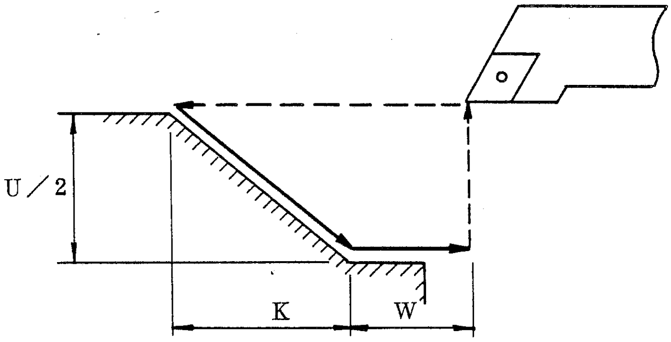
(2) The taper face cutting cycle is programmed by the following command:

G94 X(U) —— Z(W) —— K —— F —— ;

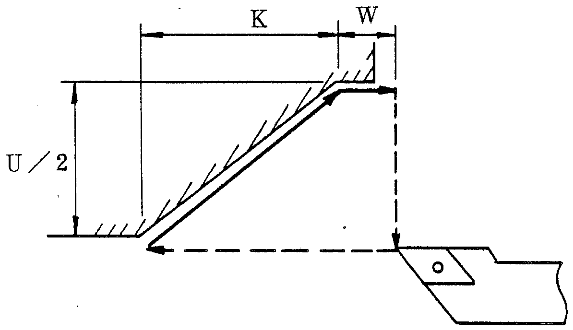


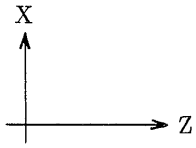
In incremental programming, the relationship between the signs of the numeric values following addresses U, W and K, and the tool paths is as follows:

(i) U < 0, W < 0, K < 0

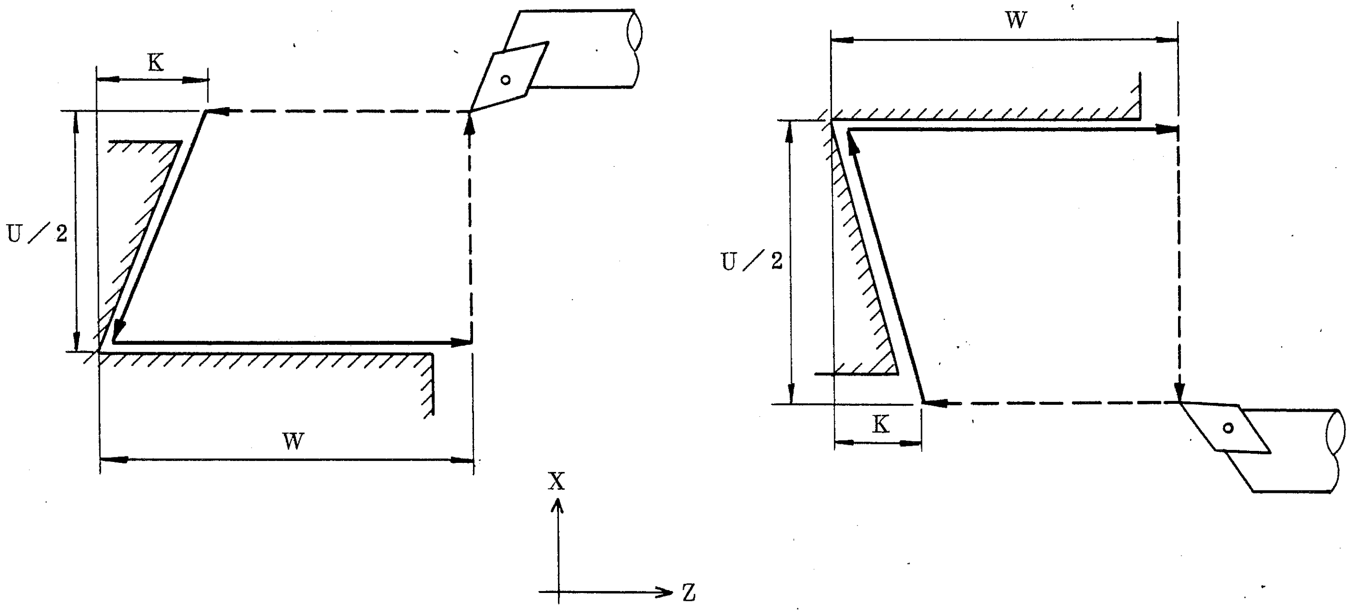


(ii) U > 0, W < 0, K < 0





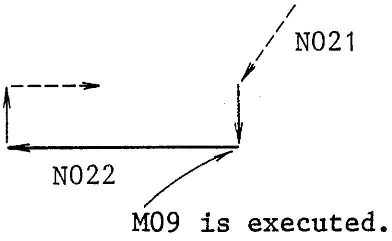
(iii) $U < 0$, $W < 0$, $K > 0$ at $|K| \leqq |W|$ (iv) $U > 0$, $W < 0$, $K > 0$ at $|K| \leqq |W|$



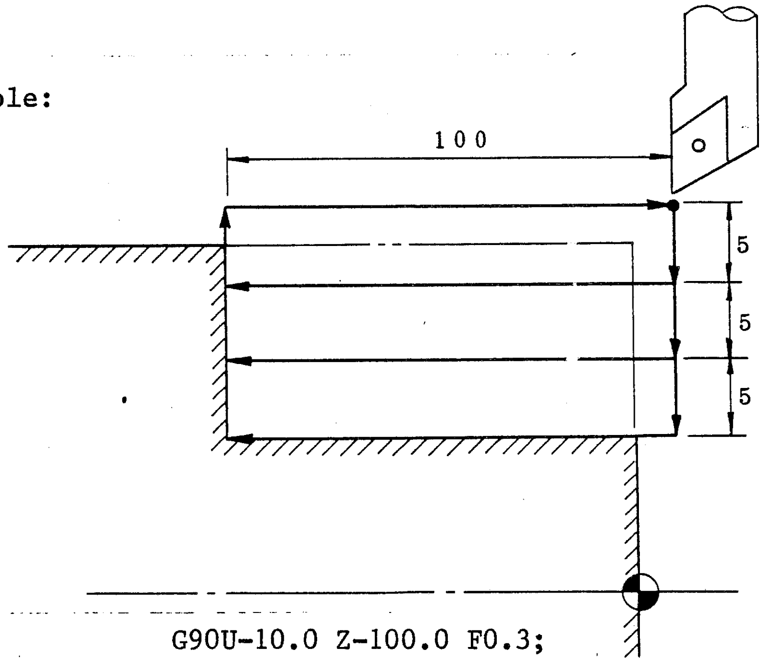
Note 1) Since data values of X(U), Z(W), I and K during a canned cycle are modal data common to G90, G92 and G94, if X(U), Z(W), I and K are not newly commanded, the previously specified data are effective.
Thus, when the Z-axis movement amount does not vary as in the example below, a canned cycle can be repeated only by specifying the movement commands for the X axis.
However, these data are cleared, if a one-shot G code (G10, G28, G30, etc.) except for G04 (dwell) or a G code in the group 01 except for G90, G92 and G94 is commanded.

Note 2) Notice the following items if M and T commands are inputted in the same block for a canned cycle.

```
N021  G00X100.Z0;
N022  G90X90.0Z-20.M09;
```



Example:



```
G90U-10.0 Z-100.0 F0.3;
   U-20.0;
   U-30.0;
```

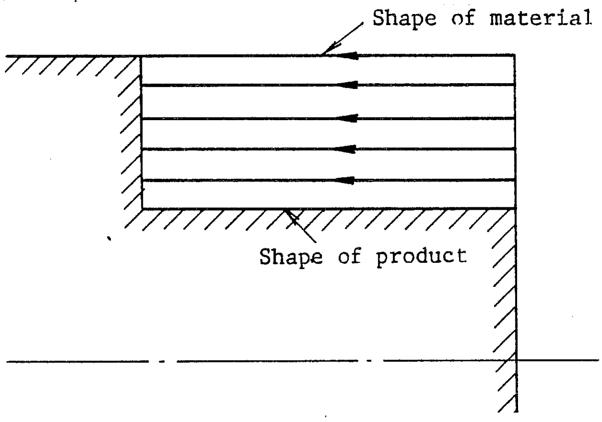
Note 1) By the use of M54 or M53, the chamfering becomes available or not available, respectively.

Note 2) When M54 for thread chamfering is provided, specify this M54 in the block before G92.
If the G92 and M54 are programmed in the same block, the chamfering may not be performed because M codes are effective after completion of axial movement.

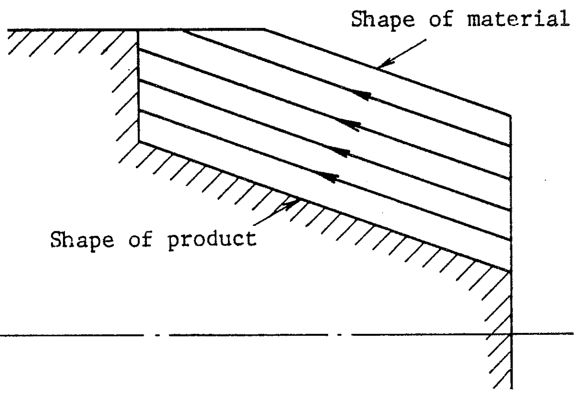
9.5 Usage of Canned Cycle

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

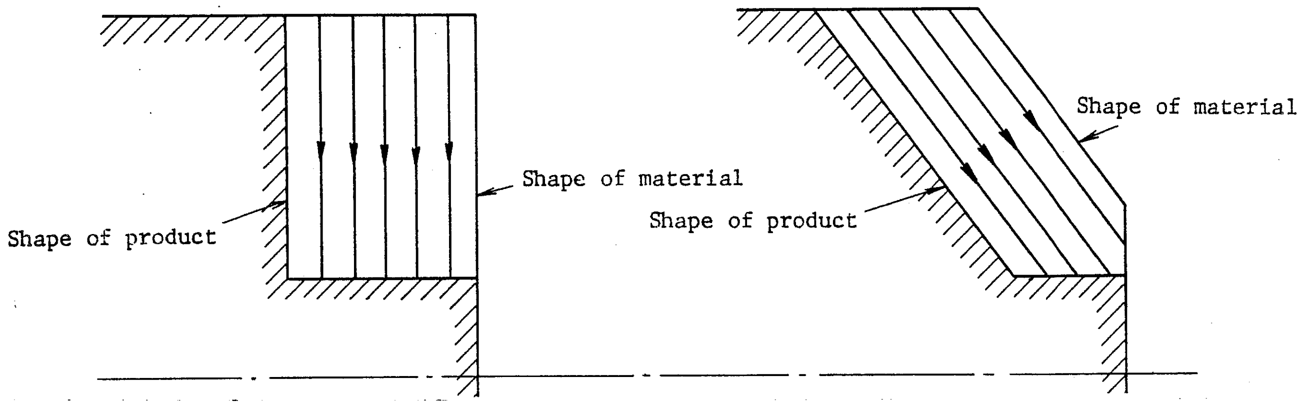
(1) Straight cutting cycle



(2) Taper cutting cycle



(3) Face cutting cycle (4) Taper face cutting cycle

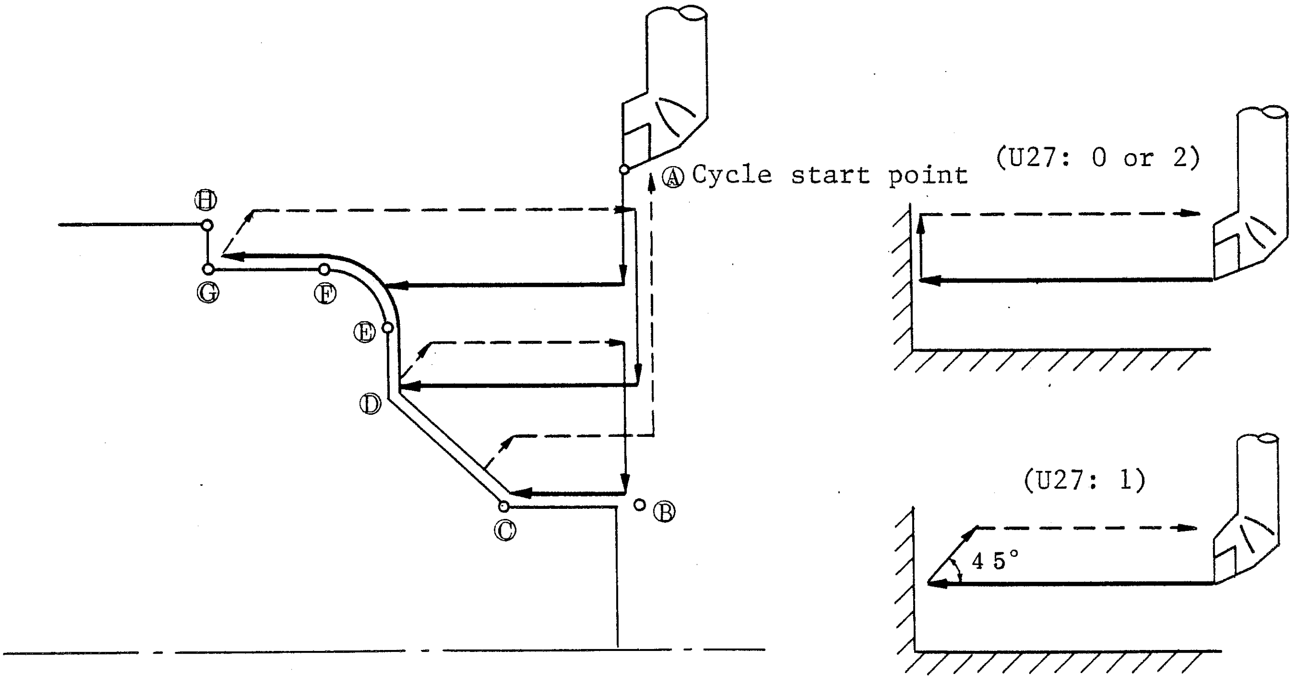


9.6 Multiple Repetitive Cycles (G70 - G76)

This function makes a programming easy. For instance, the data of the finished work shape describe the tool path for rough machining. And also, a canned cycle for thread cutting is completely performed using a one-line program.

9.7 OD Roughing Cycle (G71)

If a finished shape from (A) to (H) is given as in the figure below, the specified area is removed by cutting depth d, with finishing allowance U and W left.



Releaving a tool with an angle of 45° or chamfering in this cycle is determined by parameter U27 .
When "2" is set for U27 , the chamfering speed can be changed (refer to Parameter U70).

The command is made as follows:

```
    G71P —— Q —— U —— W —— W —— D —— F —— ;
    N o o o o~~~~~~~ ⌝
           *1 S       |
           *1 F       } Finished shape between (A) and (H)
           *1 F       |
           *1 S       |
    N x x x x ~~~~~~ ⌟
```

P : Sequence number of the first block for the finished shape

Q : Sequence number of the last block for the finished shape

U : Finishing allowance in X direction (in diameter)

W : Finishing allowance in Z direction

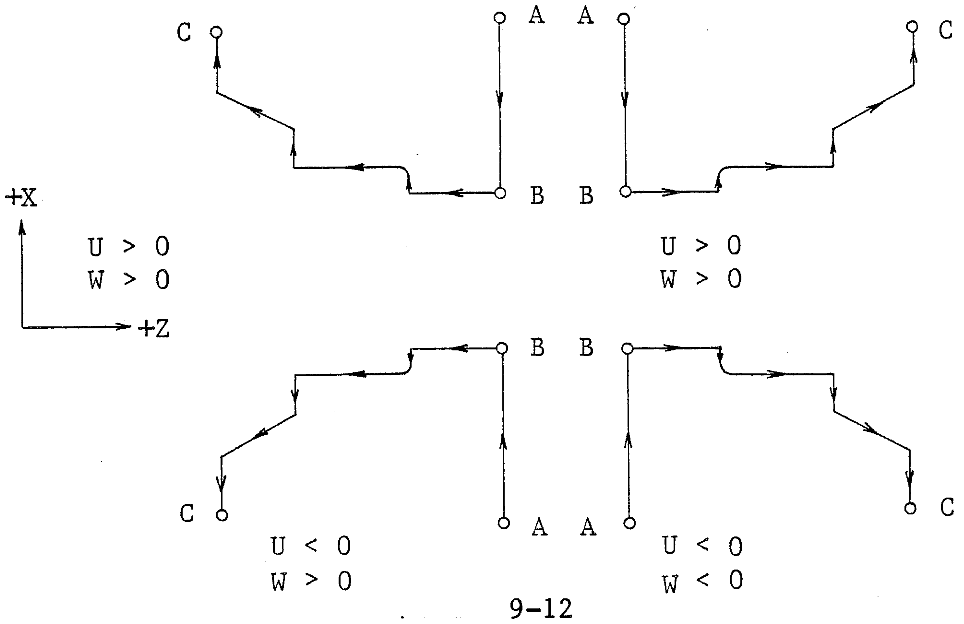
D : Cutting depth (in radius) ...{ Absolute value Decimal point input is not possible.

F : Feedrate in roughing cycle

S : Surface or spindle speed in roughing cycle

(*1) Even if F and S commands are specified between the blocks P and Q, those commands are ignored during roughing cycle, regarded as finishing cycle data.

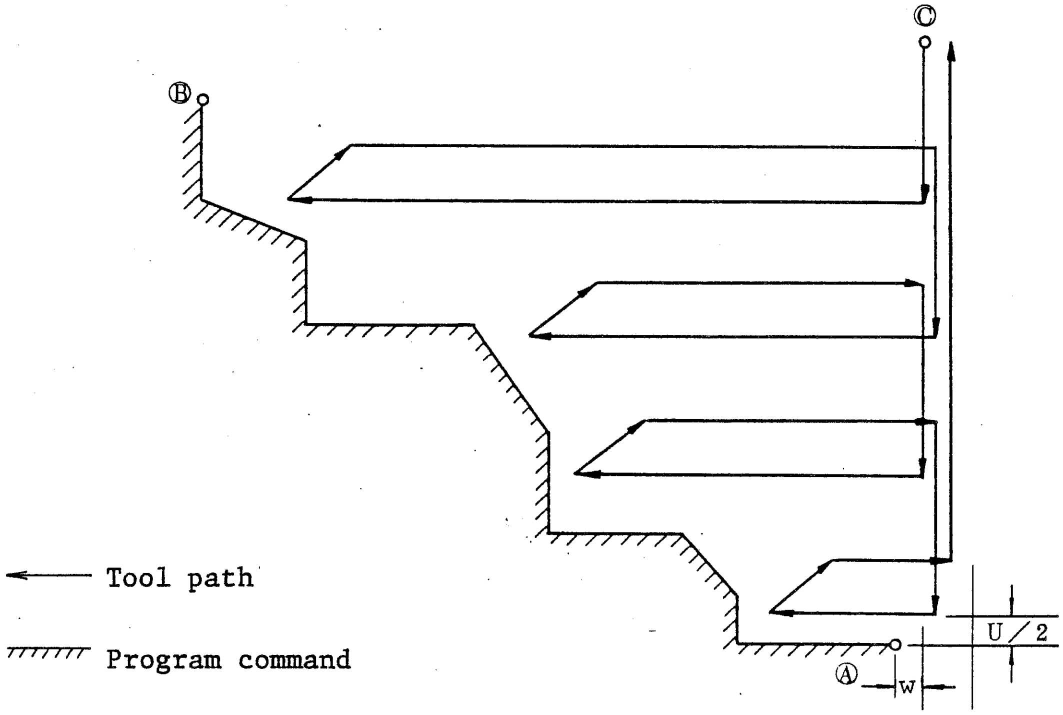
For cutting shape in the block of G71, the following four cutting patterns are considered. All of these cutting cycles are made in the Z-axis direction. The signs of finishing allowances U and W are as follows:



Note) o The tool path between (A) and (B) is specified at the sequence block of P data. A move command in the Z-axis direction cannot be programmed.

o Maximal number of concave portions on the tool path between (B) and (C) is 32.

o When the tool path between (A) and (B) is commanded by G00 or G01, cutting is performed at the rapid traverse or cutting feed rate, respectively.

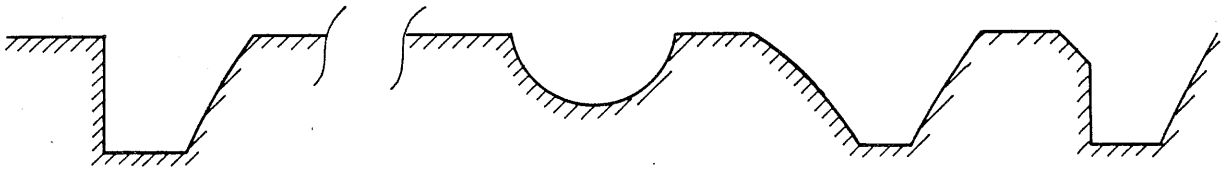


Note 1) No subprogram can be called for in the blocks between P and Q.

Note 2) Since machining is terminated referring to any of the blocks of from P to Q for a finished shape, G01 does not always come at the end but G02 or G03 may come. Set another G code after completion of this cycle.

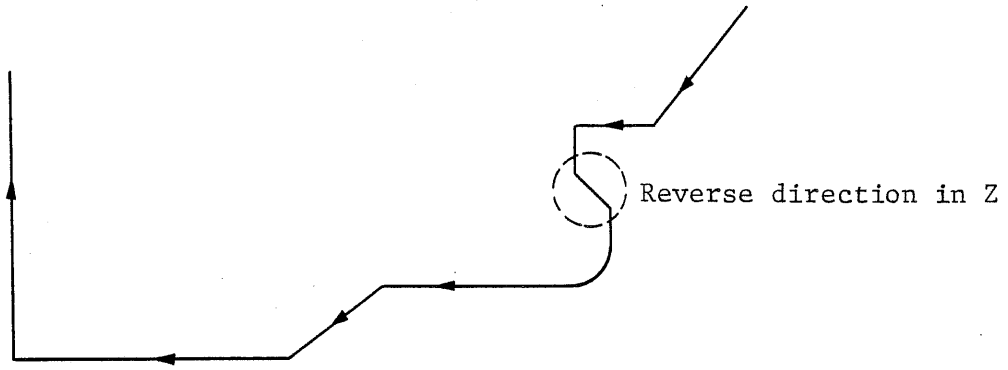
Note 3) The shape needs not to be of monotonous increase in the X direction.

And up to 32 concave portions are allowed.

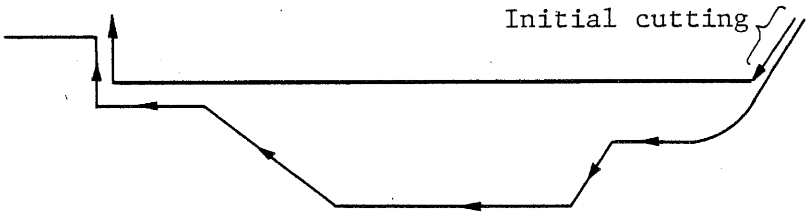


Up to 32 portions

However, the shape must be of monotonous increase in the Z direction. The following shape is not allowed and will result in alarm "REV. SHAPE CONTOUR G71 - G73".



Note 4) The cutting start face needs not be perpendicular. Any shape is permitted if it is of monotonous increase in Z direction.



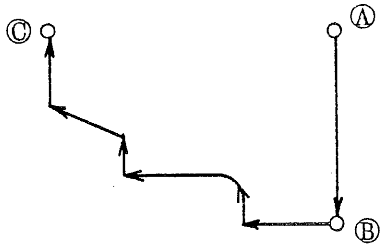
Note 5) Tool nose radius compensation value is not added to the finishing allowances U and W. And it is machined with tool nose radius compensation value being zero.

Note 6) Generally, specify W = 0 for a shape which is not of monotonous increase in X direction. If not, the tool will cut into the side of the work.

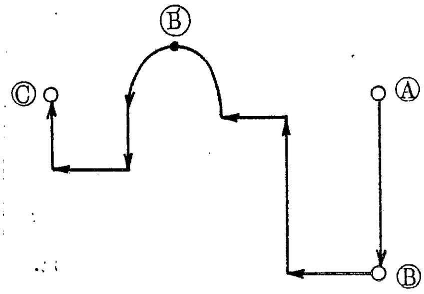
Note 7) Two axes of X(U) and Z(W) must be specified in the first block of the repetitive machining portion. If there is no Z motion in the first block, specify W0.

Note 8) Alarm No. 648 "G71, 72 LAP DISPOS. IMPOS." will be displayed if "B'" is specified when defining the machining shape between points (A), (B) and (C) when (A) (B'). The alarm is displayed case (II) but not in case (I).

(I)

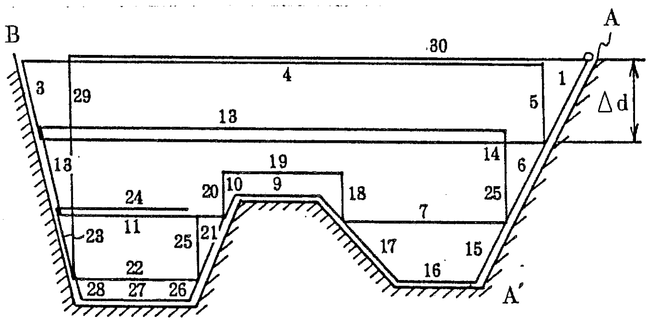


(II)



Note 9) The M-codes in P to Q sequence work in G70, not in G71, G72 and G73 cycles.

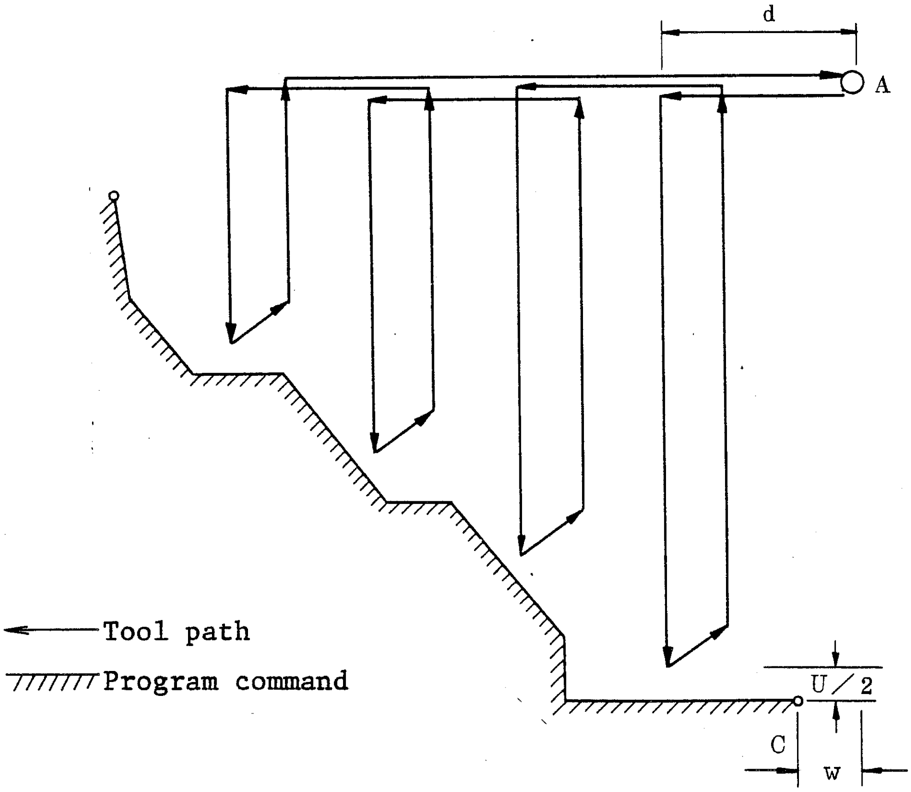
Note 10) The machining path is shown on the right.



9.8 Face Roughing Cycle (G72)

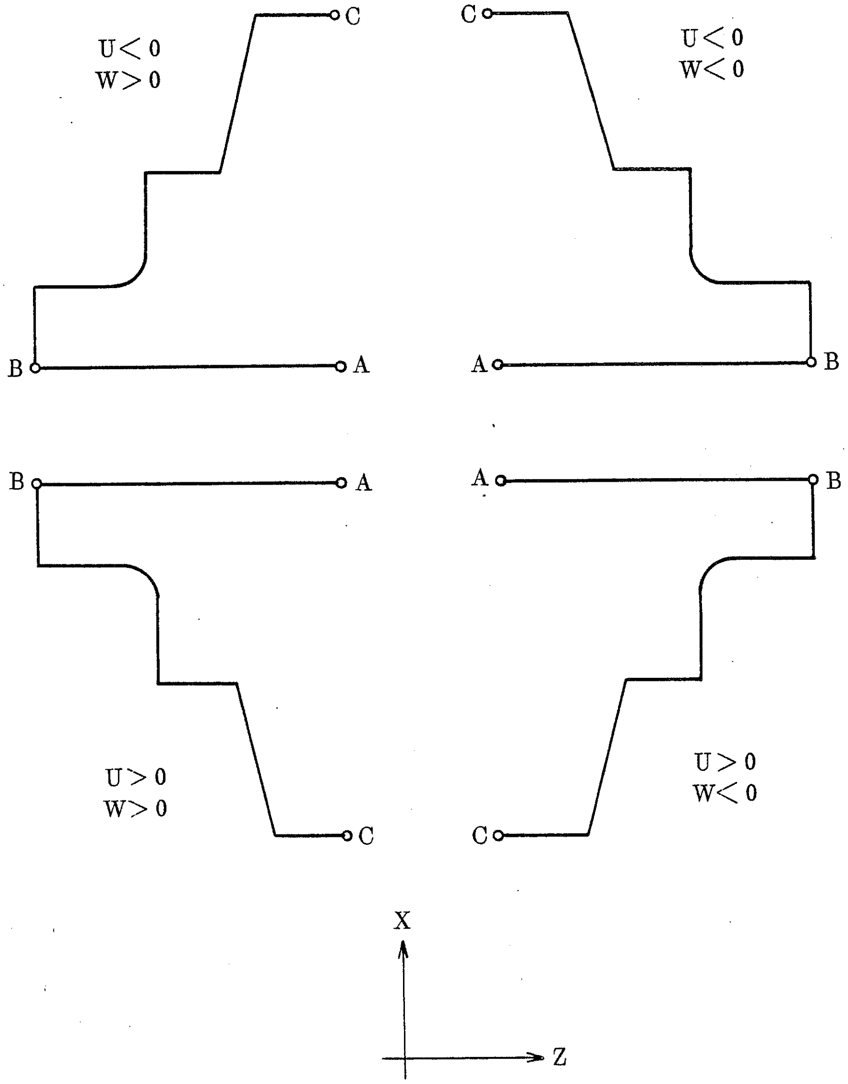
(u) (w) (d)
G72P —— Q —— U —— W —— D —— F —— S —— ;

As shown in the figure below, this cycle is the same as in G71 except that it is cut by the operation in the X-axis direction.



For G72, similarly to G71, releaving a tool with an angle of 45° or chamfering in this cycle is determined by Parameter $\boxed{\text{U27}}$.
When "2" is set for $\boxed{\text{U27}}$, the chamfering speed can be changed (refer to parameter $\boxed{\text{U70}}$).

For cutting shape in G72, the following four patterns are considered. This cutting cycle is executed by the operation in the X-axis direction. The signs of finishing allowances U and W are as follows:



Note) o The tool path between (A) and (B) is specified at the sequence block of P data. A move command in the X-axis direction cannot be programmed.

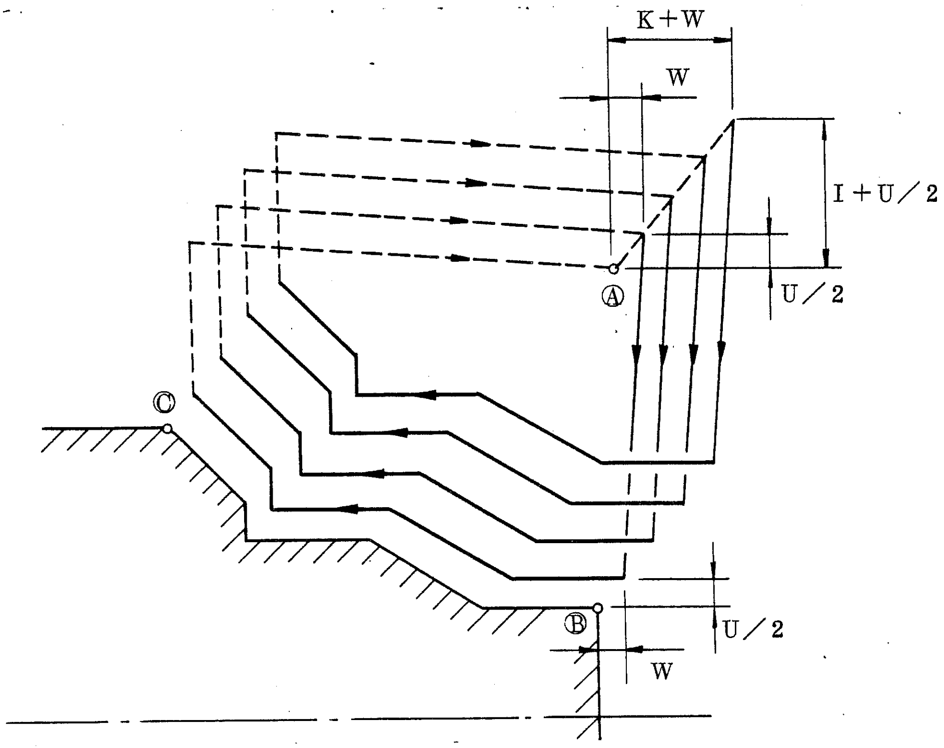
o Maximal number of concave portions on the tool path between (B) and (C) is 32.

o When the tool path between (A) and (B) is commanded by G00 or G01, cutting is performed at the rapid traverse or cutting feed rate, respectively.

The tool nose radius compensation value is not added to the finishing allowances U and W.

9.9 Closed Loop Cutting Cycle (G73)

By this cutting cycle, it is possible to efficiently perform a roughing according to a finished shape under forging or casting.



The command is made as follows:

G73P —— Q —— I —— K —— U —— W —— D —— F —— S —— ;

N o o o o

(*1)F____
(*1)S____ } Finished shape between (B) and (C)

N x x x x

P : Sequence number of the first block for the finished shape

Q : Sequence number of the last block for the finished shape

I : Distance and direction of relief in the X-axis direction ... Radius programming

K : Distance and direction of relief in the Z-axis direction

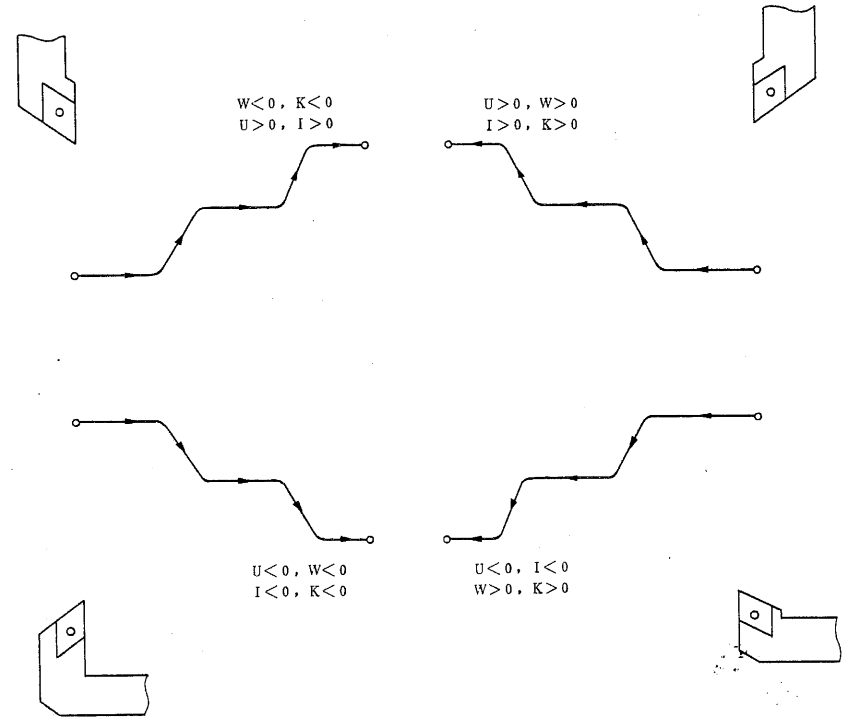
U : Finishing allowance in the X-axis direction ... Diameter programming

W : Finishing allowance in the Z-axis direction

D : Number of roughings

(*1) Even if F and S commands are specified between the blocks P and Q, those commands are ignored during roughing cycle, regarded as finishing cycle data.

For cutting shape in the block of G73, the following four cutting patterns are considered.



9.10 Finishing Cycle (G70)

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

G70P —— Q —— ;

P : Sequence number of the first block for the finished shape in roughing

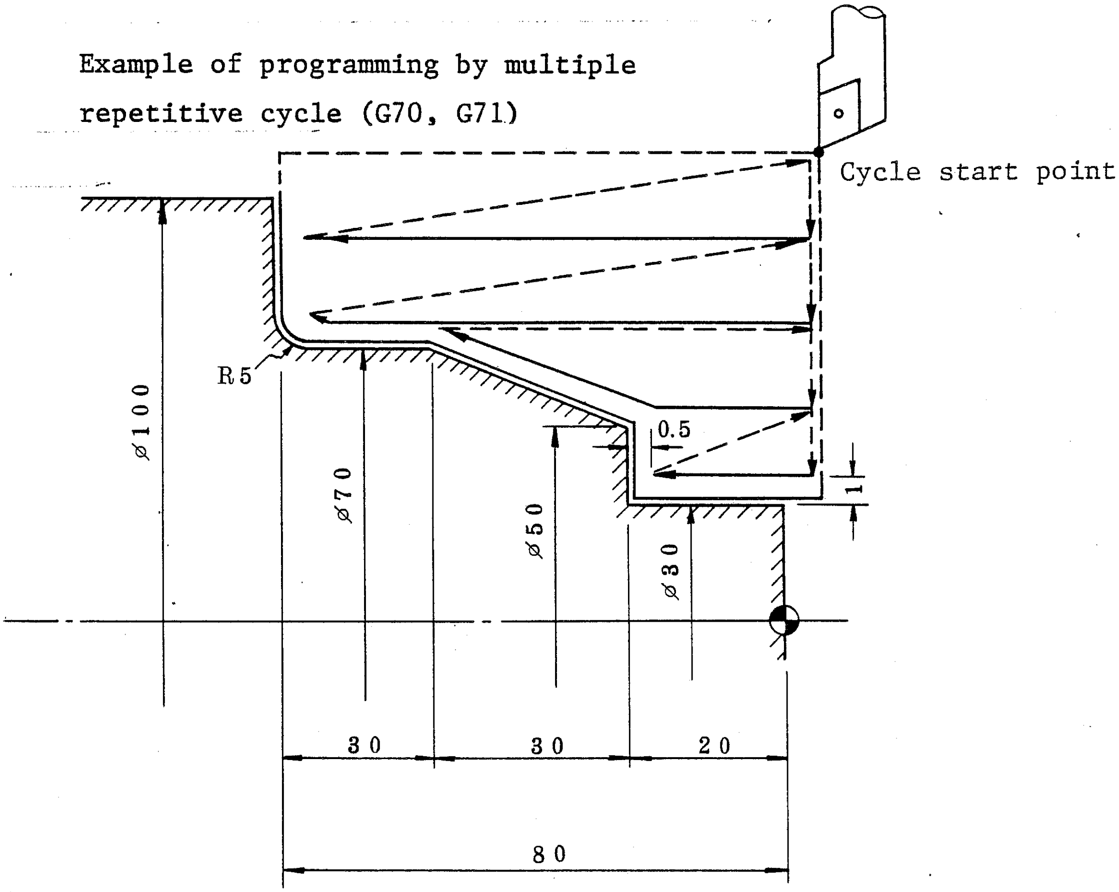
Q : Sequence number of the last block for the finished shape in roughing

Note 1) F and S functions specified in the block G71, G72 or G73 are not effective but those which are specified between the blocks P and Q are effective in G70.

Note 2) When the G70 cycle is completed, the tool returns to the starting point by rapid traverse and then machining continues from the block following the G70 cycle.

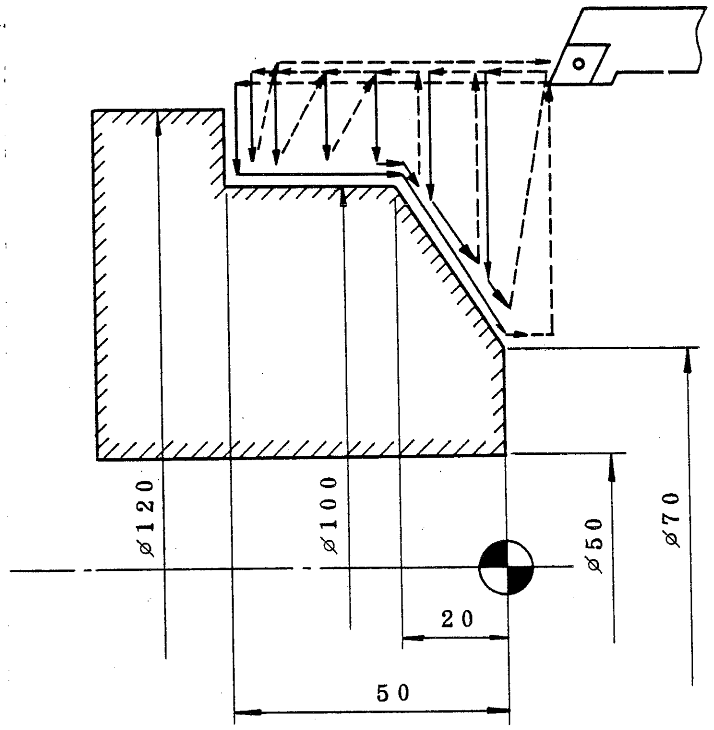
Note 3) In the blocks between P and Q used in multiple repetitive cycles (G70 - G73), the subprogram call cannot be performed.

Example of programming by multiple repetitive cycle (G70, G71)



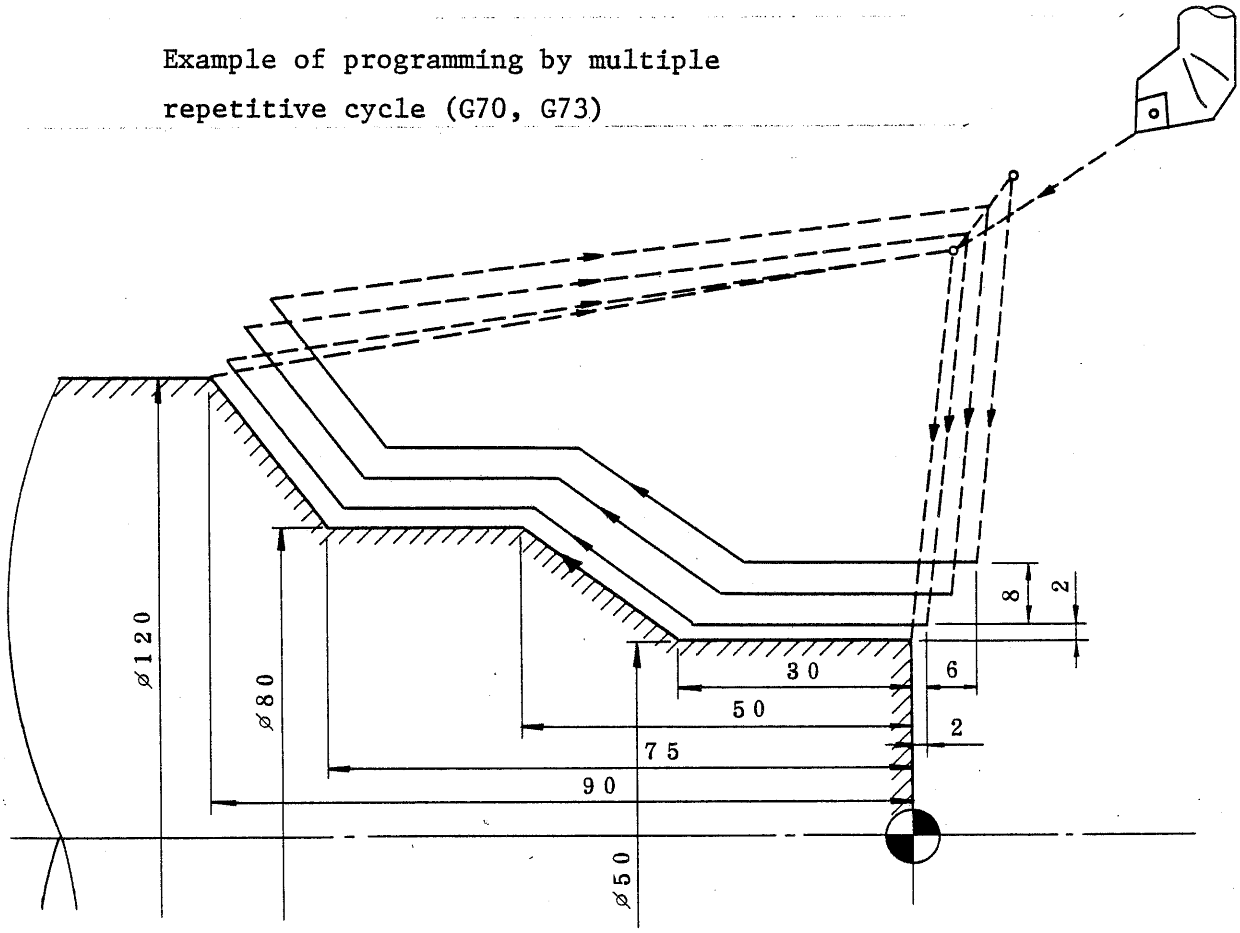
```
N010G50X300.0Z200.0;
N020G96S130;
N030M04T0101;
N040G00X110.0Z5.0; (Cycle start)
N050G71P060Q120U2.0W0.5D10000F0.35;
N060G00X30.0;
N070G01Z-20.0F0.2;
N080   X50.0;
N090   X70.0Z-50.0F0.11;
N100   Z-75.0F0.15;
N110G02X80.0Z-80.0R5.0;
N120G01X105.0;
N130G28U0W0;
N140G50X310.0Z204.0;
N150G96S200;
N160M04T0202;
N170G00X110.0Z5.0;
N180G70P060Q120;
N190G28U0W0;
```

Example of programming by multiple repetitive cycle (G70, G72)



```
N010G50X300.0Z200.0;
N020T0303M04;
N030G96S130;
N040G00X130.0Z5.0;
N050G72P060Q100U2.0W0.5D10000F0.4;
N060G00Z-50.0;
N070G01X100.0F0.2;
N080   Z-20.0F0.11;
N090   X70.0Z0F0.25;
N100   Z2.0;
N110G28U0W0;
N120G50X310.0Z205.0;
N130T0404M04;
N140G96S200;
N150G00X130.0Z5.0;
N160G70P060Q100;
N170G28U0W0;
```

Example of programming by multiple repetitive cycle (G70, G73)



```
N010G50X300.0Z200.0;
N020M04T0505;
N030G96S130;
N040G00X150.0Z5.0;
N050G73P060Q100I8.0K6.0U4.0W2.0D2F0.35;
N060G00X50.0;
N070G01Z-30.0F0.2;
N080   X80.0Z-50.0;
N090   Z-75.0F0.1;
N100   X120.0Z-90.0;
N110G28U0W0;
N120G50X305.0Z202.0;
N130M04T0606;
N140G96S200;
N150G00X150.0Z5.0;
N160G70P060Q100;
N170G28U0W0;
```

9.11 Face Cutting-off Cycle (G74)

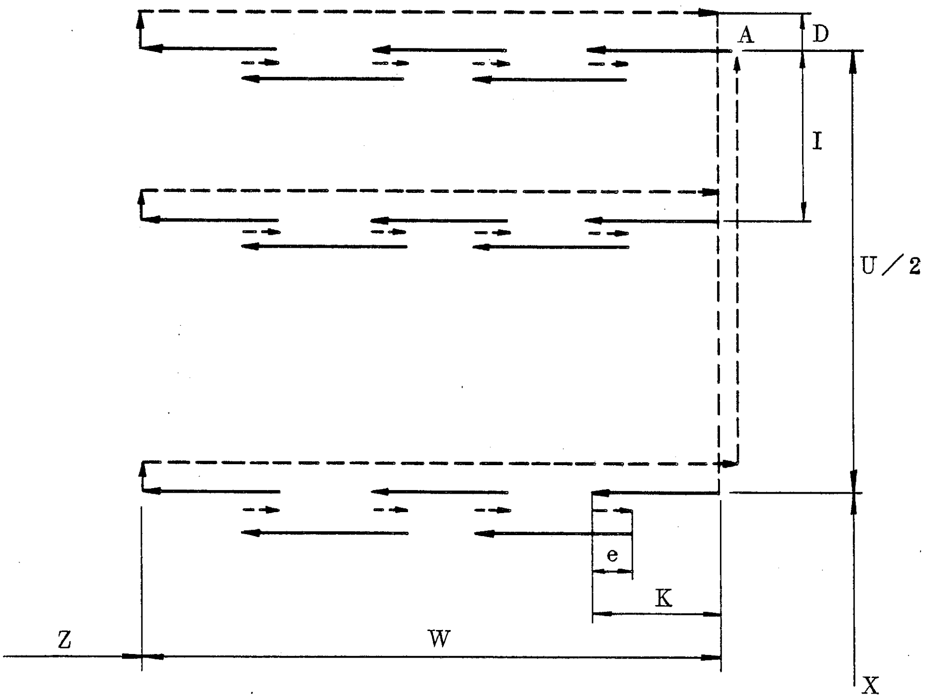
In face cutting-off, this function may be utilized for proper disposal of chips. This function also allows easy disposal of chips of hand materials such as SS which are hard to cut.

The command format is as follows:

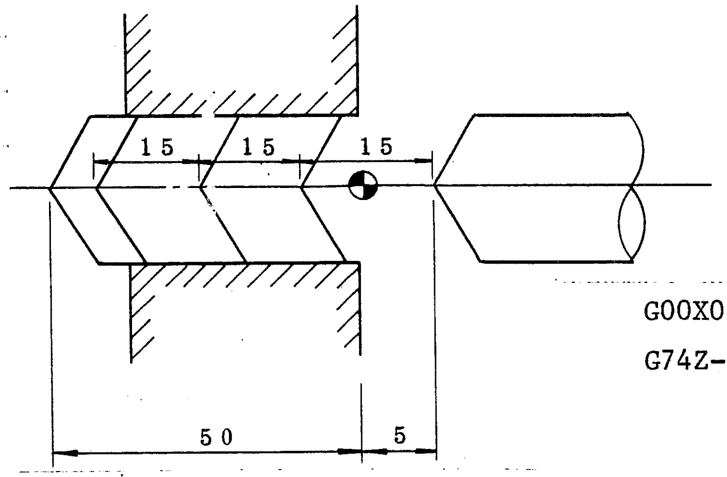
G74X(U) —— Z(W) —— I —— K —— F —— D —— ;

X :
U :
Z :
W :
} Refer to the figure below.

I : Moving amount in the X-axis direction
K : Cutting depth in the Z-axis direction
(Absolute value)
D : Relief amount in the X-axis direction
(Decimal point input is not possible.)
e : Set by parameter $\boxed{U43}$. (Relief amount in grooving)



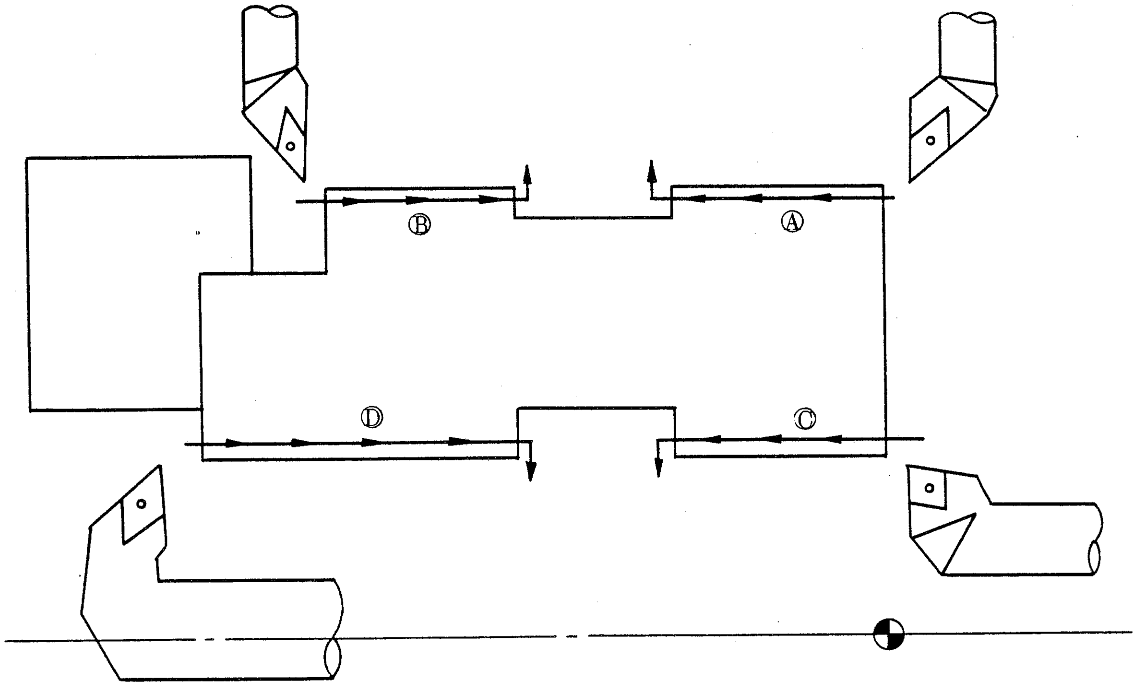
Note 1) For drilling, X(U), I and D are omitted.



```
G00X0Z5.0;
G74Z-50.0K15.0F0.2;
```

Note 2) If address D is not specified, D is assumed to be zero. The sign of D is generally positive. However, if addresses X(U) and I are omitted for O.D. or I.D. cutting, the sign of D must be specified.

For this cycle, the following four patterns are considered:



(A), (B) → D>0

(C), (D) → D<0

9.12 O.D. Cutting-off Cycle (G75)

In O.D. cutting-off, this function may be utilized to dispose of chips properly. It also allows easy disposal of chips produced by face machining.

The command format is as follows:

G75X(U) —— Z(W) —— I —— K —— F —— D —— ;

X :
U :
Z :
W :

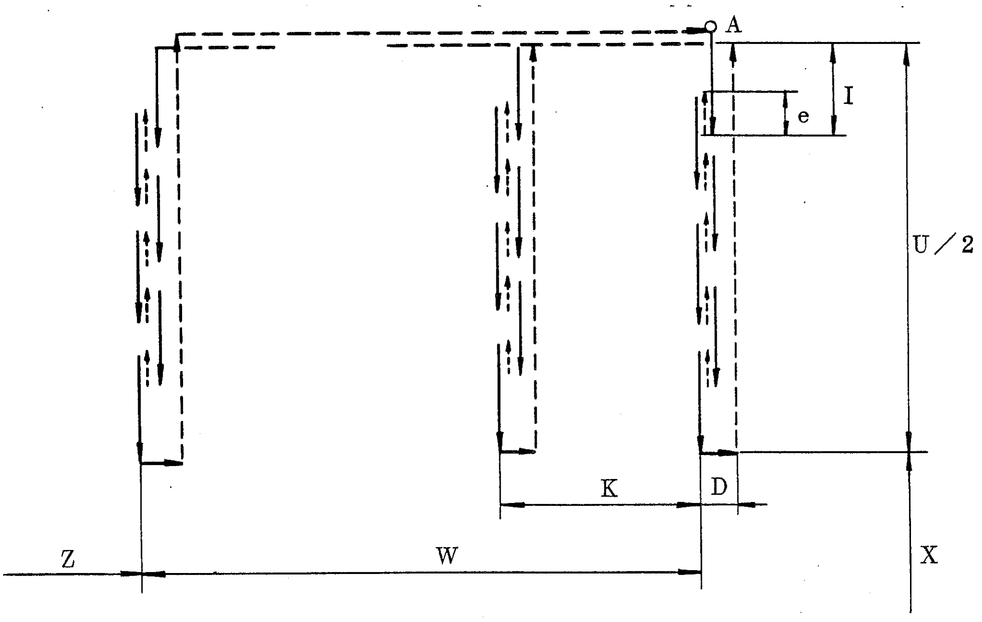
} Refer to the figure below.

I : Cutting depth in the X-axis direction
K : Moving amount in the Z-axis direction

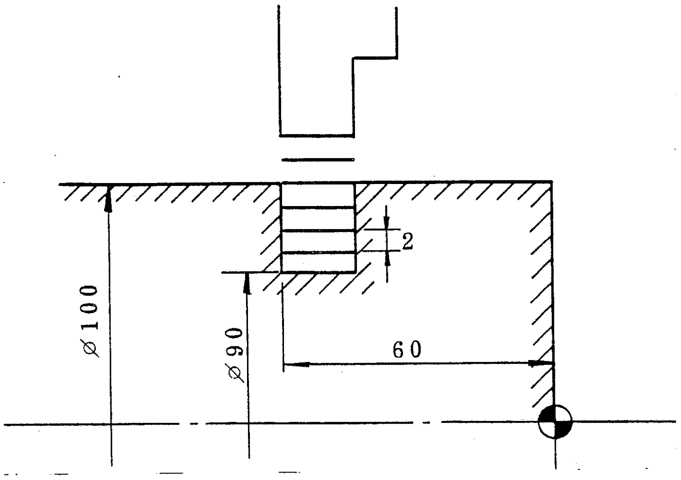
} (Absolute value)

F : Feedrate
D : Relief amount in the Z-axis direction
(Decimal point input is not possible.)
e : Set by parameter $\boxed{U43}$. (Relief amount in grooving)

By using G75 command, the cycle as shown in the figure below is programmed.



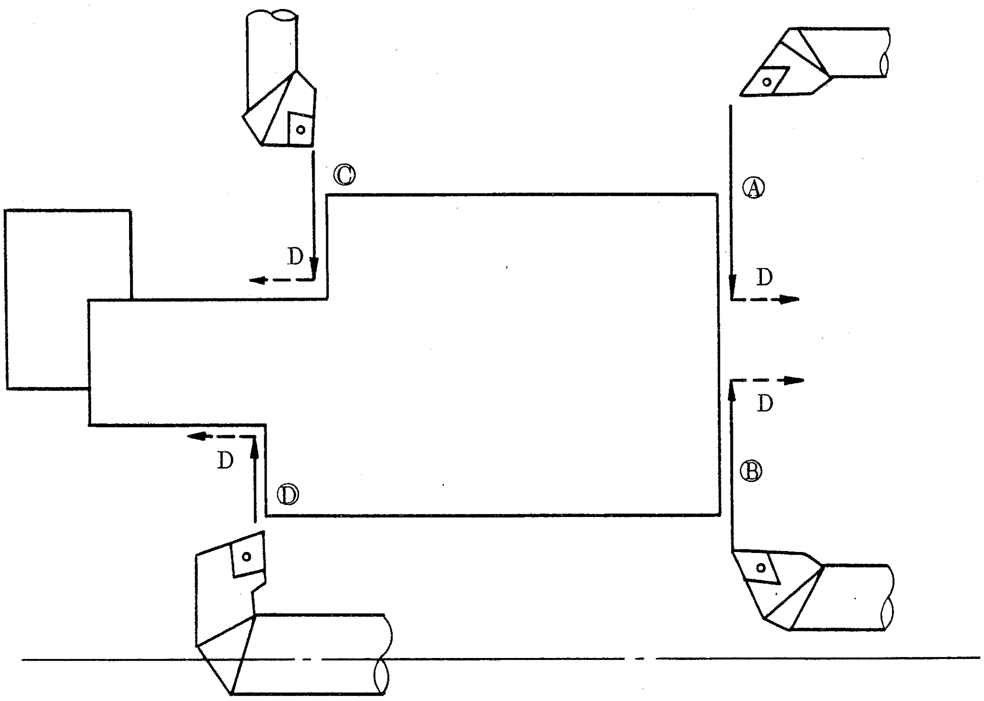
Note 1) For O.D. or I.D. grooving, addresses Z(W), K and D are



```
G00X105.0Z-60.0;
G75X90.0I2.0F0.05;
```

Note 2) If address D is not specified, D is assumed to be zero. The sign of D is generally positive. However, if addresses Z(W) and K are omitted for face cutting, the sign of D must be specified.

For this cycle, the following four patterns are considered:

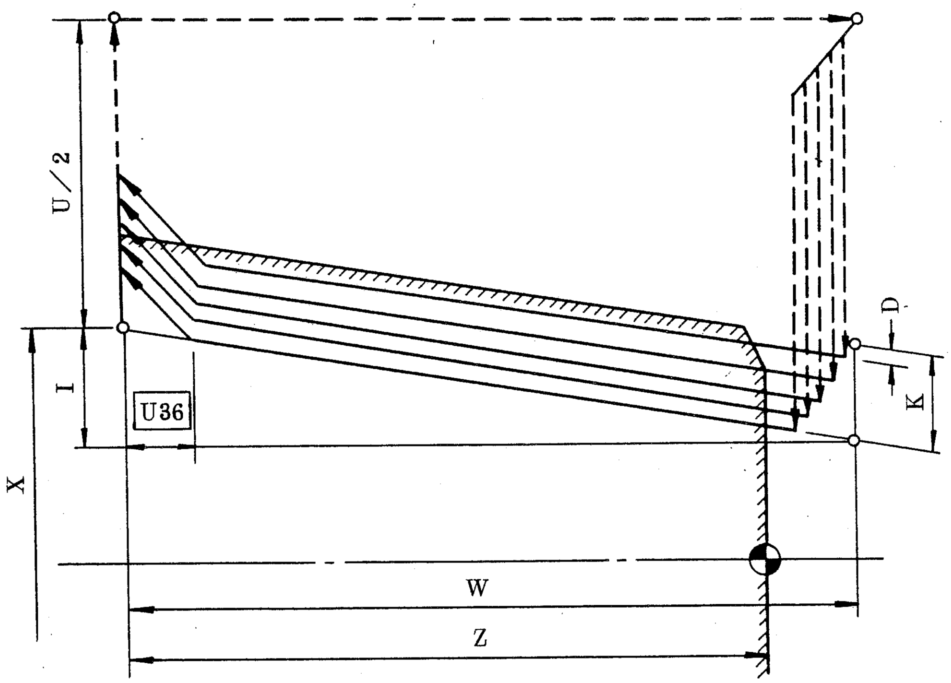


(A), (B) → D>0

(C), (D) → D>0

9.13 Thread Cutting Cycle (G76)

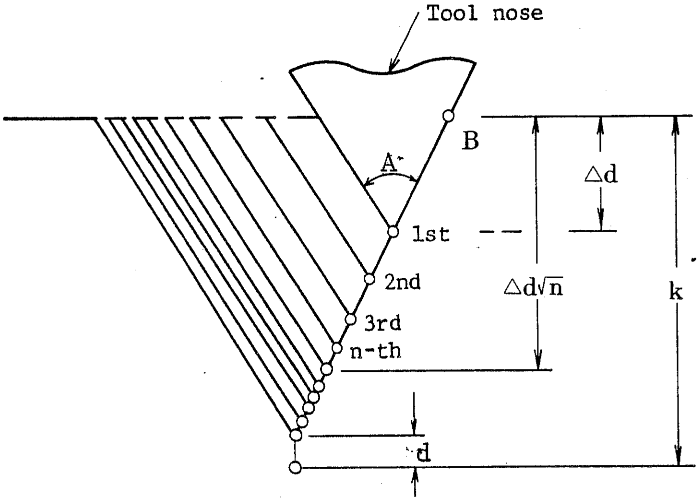
Thread cutting cycle as seen in the figure below is programmed by the G76 command.



Specified by F or E code

Rapid traverse

U36 ; Thread chamfering amount
(parameter)



$$d = \frac{\boxed{U33}}{2}$$

U33 : Threading finish allowance (parameter)
(diameter value)

The command format is as follows:

$$\text{G76X(U) —— Z(W) —— I —— K —— D ——}\left\{\begin{array}{l}\text{F —— A —— ;}\\ \text{E ——}\end{array}\right.$$

X : Thread bottom diameter in the thread end point

U : Moving amount in the X-axis direction from cycle start point to end point (thread bottom)

Z : Z coordinate value in the thread end point

W : Moving amount in the Z-axis direction from start point to end point

I : Difference between thread radii in the starting and end points (Generally, I 0 for O.D. and I 0 for I.D.)

K : Height of thread (specified by radius value)

F,E: Thread lead

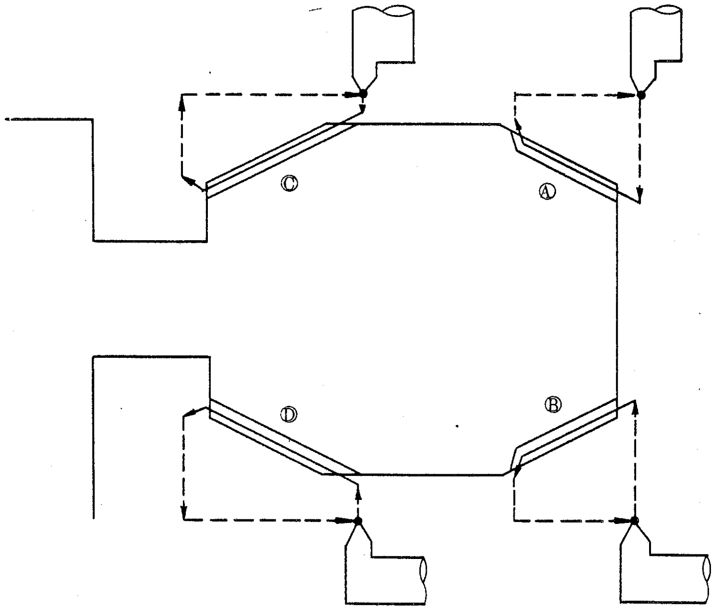
A : Angle of tool nose (This angle is data for one-edge cutting and can be set from 0° to 120° in 1° increment.)

D : Cutting depth in first cutting specified by radius value (Decimal point input is not possible.)

When the thread lead is expressed as L, the chamfering amount can be set from 0.1L to 4.0L in 0.1L increment by parameter $\boxed{\text{U36}}$.

Each cutting depth is fixed by expressing it as D for the first time and as $D\sqrt{n}$ for n-th time. Cutting depth, however, can be clamped by parameter $\boxed{\text{U62}}$ setting value lest it should become too small.

For this cycle G76, the following four patterns are considered:



(A), (D) → I < 0

(B), (C) → I > 0

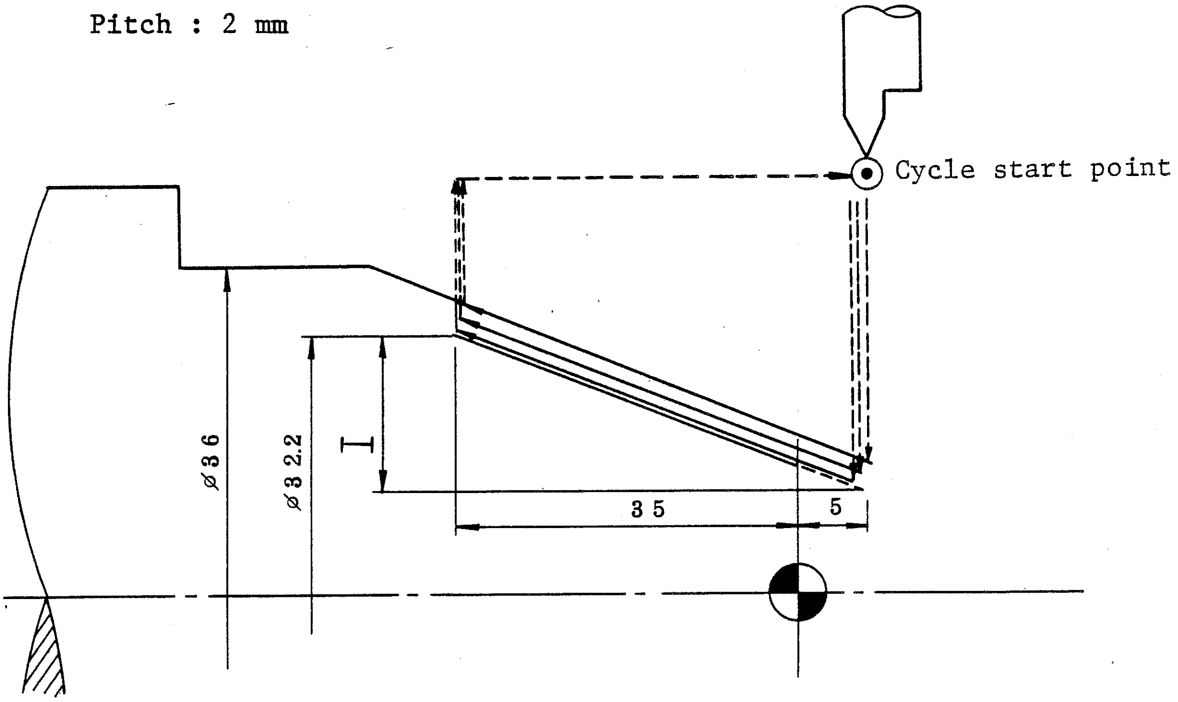
←——Specified by F or E code

←----Rapid traverse

Example of programming by multiple repetitive cycle (G76)

Taper : 1/16

Pitch : 2 mm



```
N010G50X300.0Z200.0;
N020M03T0101;
N030G00X40.0Z5.0; (Cycle start)
N040G76X32.2Z-35.0I-1.25K1.246D200A60F2.0;
```

Note 1) Up to fourth lower digit than decimal point can be inputted for E code (in mm).

9.14 Note on Multiple Repetitive Cycles (G70 - G76)

(1) Within the finished shape blocks specified by P and Q in G70, G71, G72 or G73, the following commands cannot be specified.

- M98/M99
- T code
- G20, G21, G52, G53, G68, G69, G94 and G95
- G32, G33, G77, G78 and G79
- G10, G27, G28, G29 and G30

(2) When G70, G71, G72 or G73 is executed, the sequence numbers specified by P and Q must not be the same as those which have already been stored in the program.

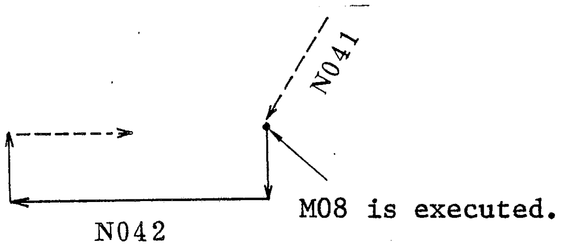
(3) When the last command in the blocks specified between P and Q in G70, G71, G72 or G73 includes chamfering (G01X —— I —— ;) or (G01Z —— K —— ;), or corner R (G01Z —— R —— ;) or (G01X —— R —— ;), alarm 621 "ILLEGAL NEXT BLOCK" will be displayed.

(4) When G71, G72 or G73 is specified, G00 or G01 must be in effect in the block in which the sequence number is specified by P.

(5) While a multiple repetitive cycle (G70 - G76) is being executed, it is possible to stop the cycle by pressing the [O] (FEED HOLD) key and perform a handle interruption (shift the tool position in the manual operation mode). But the cycle must be restarted from the position where the manual operation had begun, by pressing the [I] (CYCLE START) key. If the cycle restarts without returning the tool to its original position when handle interruption was started, the tool path is shifted by the amount of the movement in the manual operation.
The amount of the movement in the manual operation is cancelled by pressing the [//] (RESET) key.

(6) When M and T commands are inputted in the same block for G70-G76, notice the position where M08 is executed.

```
N041  G00X100.Z0;
N042  G71P101Q103U0.5W0.5
         D4000F0.5S150M08;
         .
         .
         .
         .
N101  G01X90.F0.5;
N102     Z-20.;
N103     X100.;
```



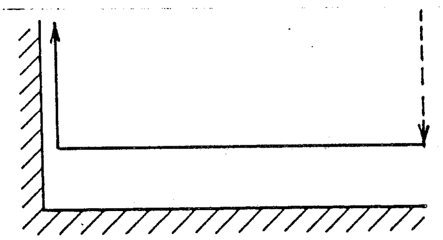
Supplement:

Setting parameters $\boxed{U27}$ and $\boxed{U70}$:

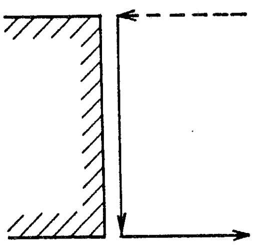
The rest interval after turning in complex fixed cycles G71 and G72 can be changed using parameter $\boxed{U27}$.

When $\boxed{U27}$ = 0,

G71

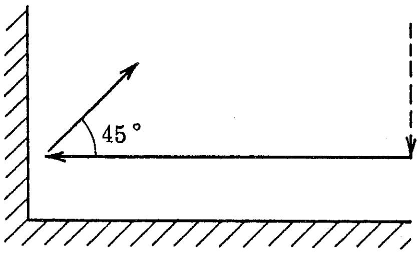


G72

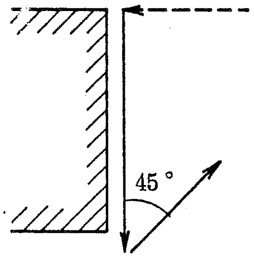


When $\boxed{\text{U27}}$ = 1,

G71

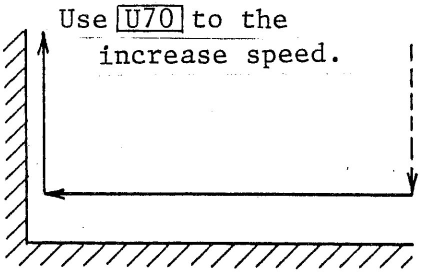


G72



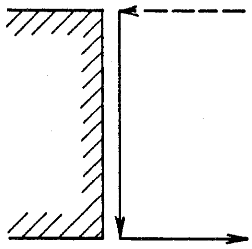
When $\boxed{\text{U27}}$ = 2,

G71



Use $\boxed{\text{U70}}$ to the increase speed.

G72



Use $\boxed{\text{U70}}$ to the increase speed.

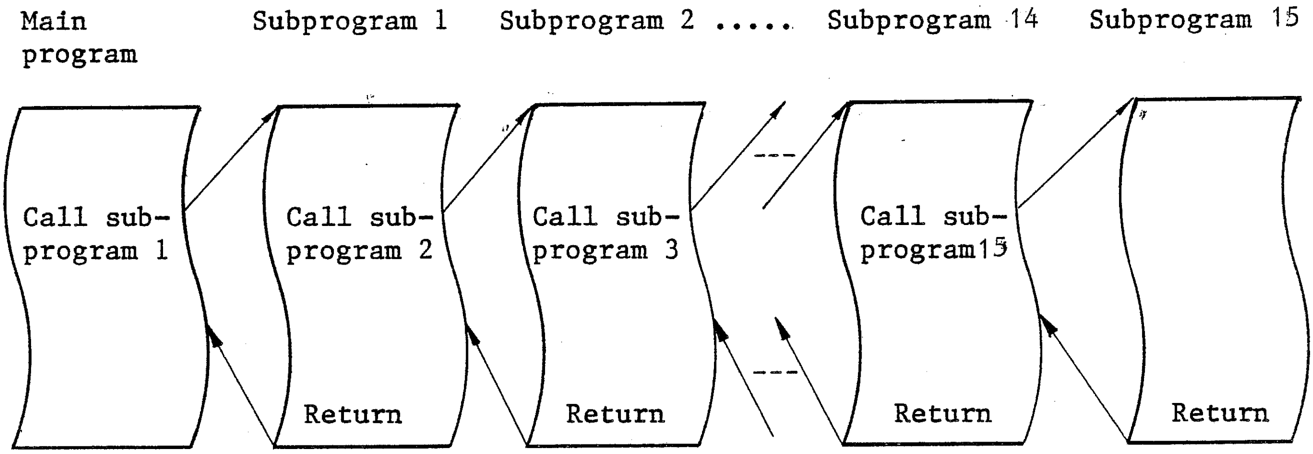
Setting parameter $\boxed{\text{U70}}$ the following equation is used to determine the speed increase.

F2 = F1* $\boxed{\text{U70}}$ /10

$\boxed{\text{U70}}$ =0 is read 10.

10. SUBPROGRAM

When a program contains certain fixed sequences such as coordinate measurement or frequently repeated patterns, these sequences or patterns can be entered into the memory as a subprogram. This can simplify programming.
The subprogram is called for by the main program.
A subprogram can call for antother subprogram.
Subprogram call by the main program is called nesting. The nesting can be performed up to 15 times as shown below.



A call command can call a subprogram repeatedly. A call command can command the repeating of a subprogram up to 9999 times.

10.1 Preparation of Subprogram

A subprogram is prepared in the following format:

```
........................;
........................;
       .
       .
       .
       .
........................;
M99;
```

Subprogram end command M99 need not be commanded in a block by itself.

Example:

```
X ...........M99;
```

10.2 Execution of Subprogram

A subprogram is executed by the call of main program or another subprogram.
A subprogram is called for by the following format:

```
M98P.........L.........;
         |         |
         |         |____Number of times the subprogram is to
         |              be repeated.
         |
         |______________Subprogram No.
```

When L is omitted, the subprogram is repeated once.

Example: M98P0100L3;
This command is read "Call the subprogram 100 and repeat it three times".

The subprogram call command (M98P____L____) and a move command can be commanded in the same block.

Example: X1000M98P1200;
In this case, after completing the motion in X-axis direction, the subprogram (subprogram number 1200) is called.

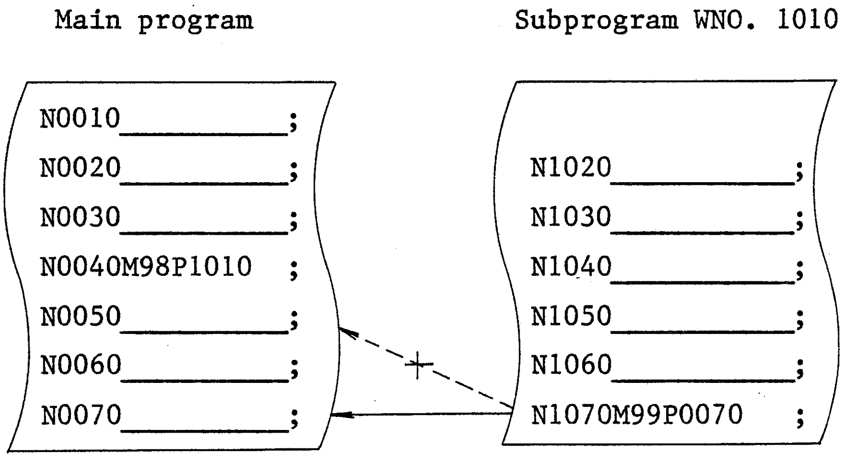
When the subprogram is called for by another subprogram, the sequence executed is the same as in call by the main program.
Up to 15 other subprograms can be called by a subprogram.

Note) If the subprogram number specified by address P can not be found, an alarm is displayed.

10.3 Special Uses

The following special uses are available.

(1) when a sequence number is specified by the address P at the last block of a subprogram, the control does not return to the block after the block called by the main program, but rather to the block with the sequence number specified by the address P.



Main program

```
N0010__________;
N0020__________;
N0030__________;
N0040M98P1010  ;
N0050__________;
N0060__________;
N0070__________;
```

Subprogram WNO. 1010

```
N1020__________;
N1030__________;
N1040__________;
N1050__________;
N1060__________;
N1070M99P0070  ;
```

11. SUPPLEMENT

11.1 Check Function

Used to check whether the program has been prepared correctly before being run in automatic mode. The tool path is displayed on the CHECK picture.

Steps

1 Select PROGRAM picture and the press CHECK key. CHECK picture will then be displayed.

2 (1) Press the CHECK CONTINUE key when the check of the tool path is to be continued. The tool path will then be indicated by a dotted white line for quick feed or by a solid white line for machining feed. When setting the coordinates, no line is drawn because of point displacement.

The menu display is inverted during preparation and is re-inverted when this is completed.

(2) Press the CHECK STEP key to check the tool path intermittently.

(3) Press the STORE key when saving the tool path.
The tool path remains on the picture after changed to another picture when the CHECK picture is displayed again.

(4) Press the TOOLPATH ERASE key to erase the tool path.

(5) Press the SCALE key to change the scale. The carousel will then be displayed. Position it using the carousel shift key and set the scale length using ten keys.

End

Note 1) Graphic lines are displayed regardless the parameter P15 value.

Note 2) This differs from the MAZATROL program since the graphic-check and the simulation functions are not provided.

Note 3) The tool path starting point can be fixed using parameters U76 (X) and U77 (Z). C-axis: 0

CHECK picture:

BLOCK SKIP is possible when EIA/ISO programs are being checked. Select the CHECK picture, press the (Menu selection) key and select the skip number.

BLOCK SKIP 1	BLOCK SKIP 2	BLOCK SKIP 3	BLOCK SKIP 4	BLOCK SKIP 5	BLOCK SKIP 6	BLOCK SKIP 7	BLOCK SKIP 8	BLOCK SKIP 9

The selected BLOCK SKIP function is valid only for checking. When running the program, therefore, input skip number on the POSITION , COMMAND and TRACE pictures.

11.2 Restarting

To restart the machining at an intermediate program point after RESET an emergency-stop after a tool breaks, use the following procedure.

11.2.1 Procedure

Steps

1 Check the restart position.
Check it the turret interferes with workpiece when moving to the starting point. If interference occurs, move the turret manually.

2 Select any POSITION, COMMAND or TRACE display.

3 Put the M (RESTART) key. The restart menu will appear.

(1) Enter the starting sequence number. (Press the N key and use ten keys.)

(2) Press the SEARCH START key. A search will then start at the top of the program. The display will be inverted during search and will be reset to the initial position the search is completed.

(3) Press the SEARCH STEP key when starting a run a step ahead of the search block. Steps are reset after each search, but are not reversible.

Starting a run at a block without a sequence number.

```
Example)  N006  G00X200.Z100.;
                G00X100.Z2.;
                G01X95.F0.5; <------Start of run
                G01Z0;
          N007  G01Z-50.;
```

(i) Sequence No. setting

(ii) Search with the SEARCH START key.

(iii) Search repeat with the SEARCH STEP key.

The SEARCH STEP function does display the sequence, so it should not be used when the search function is not understood.

4

Use the M key when selecting an M-code. Menu key or ten keys can be used.

5

Press the ⎕ (CYCLE START) key to start the machining operation.

End

Note 1) Correct positioning cannot be performed if G50 and X and Z are not specified (incremental program) before the search block.

Note 2) During search, keep correcting the data (tool offsetting, nose radius and nose point) to same values as for machining.

Note 3) No corrections are made in following cases.

1) Program mirror image (G68) is used.

2) Machine lock is cancelled after the search.

3) Manual interrupt occurs during the move to the restart position.

4) Blocks are searched when X or Z absolute commands are not given after the skip command (G31).

5) Programs in which the G50 command is given when X or Z absolute commands are not given after skip command (G31).

Note 4) Complex fixed cycle (G70 ...) blocks cannot be searched.

Note 5) Alarm No. 628 "G30 START COND. ERROR" will be displayed if the G27 command is given during a search started without a zero-point reset after turning the power on or resetting after an emergency stop.

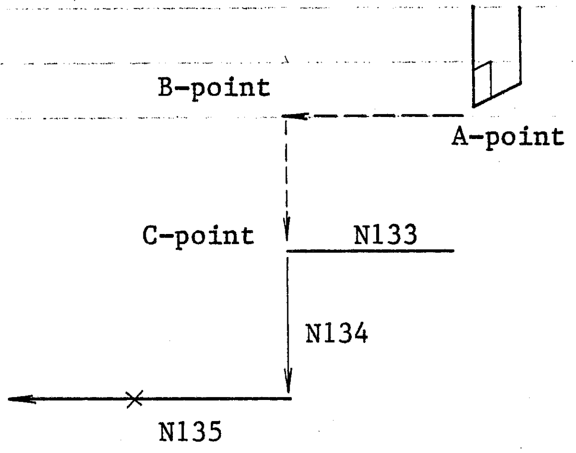
Note 6) After the restarted search, alarm No. 488 "OPERATION NOT ALLOWED" will be displayed if an attempt is made to change the program data. Changes can be made when the position for completing the restarted search is cancelled by any of the following.

i) Setting the mode to other than RESTART
ii) Completion of restarted operation
iii) Resetting during restarted operation

Note 7) For programs to be started at a fixed position, this will be moved if started at other position.

```
Example)  G00 G96 G99 M39 S130;
          G50X300.Z300.S2000; ←——G50 command is
                      .             given although
                      .             no move is made
                      .             to a fixed
                                    position.
```

11.2.2 Operation



When machining is restarted at the N134 command after an emergency stop during N135 machining, the cause of the emergency stop must be cleared and the X- and Z-axis zero-points reset.

If interference can occur during restart from the zero-point, first move the axis to safe position.

In this case, follow the above procedure to search for N134. When the □ (CYCLE START) key is pressed after the search ends, the process restarts as follows:

(1) The M-codes set to the A-point are run one at a time.
(2) A-point data are reset.
  i) Setting coordinate systems (search blocks)
  ii) Renewal of G-mode (G00/G01, G40/G41/G42, G52/G53, G96/G97, G98/G99)
  iii) Renewal of F- and S-data
  iv) Renewal of tool offset data (G10 command is given until NC inside buffering block is reached in the restarted search.)

(3) The tool is changed at the A-point.

(4) Z- and C-axes are quick-feed to the B-point one step before the search block. Tip radius compensation will then be applied.

(5) X-axis is quick- or cutting-feed to the C-point one step before search block. Either G00 or G01 is used for this feed depending on the G-mode condition.

(6) Search sequence N134 is run.

Note 1) When restarting the machine with the ATC, move to its position, restart the search and press the ⎕ (CYCLE START) key.

Note 2) Machining is performed at parameter $\boxed{\text{S15}}$ speed unless F-data are input during the block search in the G01 mode.

Note 3) Single step stop is possible at C-point using parameter $\boxed{\text{U17}}$.

11.3 End Processing

If any M02, M30, M198, M199 or EOR (%) is input, the numerical controller will performs the end process. The contrary occurs with M98 and M99.

(1) M02, M30

Used only for tool life processing.

(2) M198, M199

Used for tool life processing, parts count and work number search.

```
M198(199)  P111  Q1 ;
```

Q1 — Specification of parts count (renewal of position-command counter)

0: no parts count

1: parts count

P111 — Next work number specified.

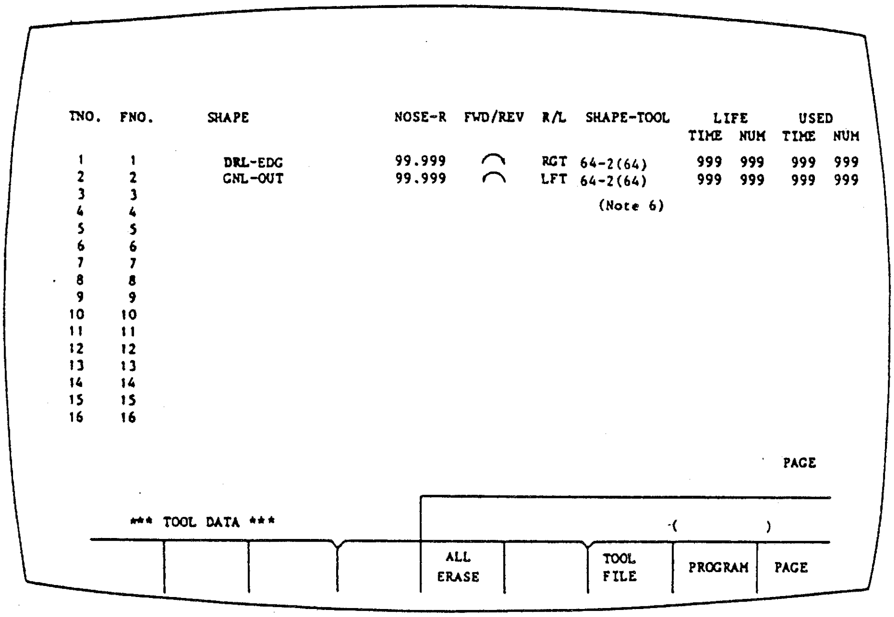
M198(199) — M-codes for program sequence

M198: continuous operation after parts count and work number search

M199: parts count and work number search are performed and completed.

11.4 Tool Life

A life-end alarm is given and the automatic machining operation stops when the wear/time tool data on the TOOL DATA picture indicate that the tool has reached the end of its wear/time.



```
TNO.  FNO.       SHAPE              NOSE-R  FWD/REV  R/L  SHAPE-TOOL      LIFE       USED
                                                                       TIME  NUM  TIME  NUM
  1     1        DRL-EDG            99.999    ⌒      RGT  64-2(64)      999   999  999   999
  2     2        GNL-OUT            99.999    ⌒      LFT  64-2(64)      999   999  999   999
  3     3                                                   (Note 6)
  4     4
  5     5
  6     6
  7     7
  8     8
  9     9
 10    10
 11    11
 12    12
 13    13
 14    14
 15    15
 16    16
                                                                                  PAGE

        *** TOOL DATA ***                                              (         )
```

Service life: alarm No. 151 "TOOL LIFE TIME ERROR (NUMBER)"

Time life: alarm No. 152 "TOOL LIFE TIME ERROR (TIME)"

11.5 Spare Tools

Wear and time spare tools are selectable using parameter $\boxed{P4}$.

```
TNO. ............ SPARE-TOOL

 1                 11-2(51)
 .                    .
 .                    .
 .                    .
```

P4 ... 0: no spare tool indexed

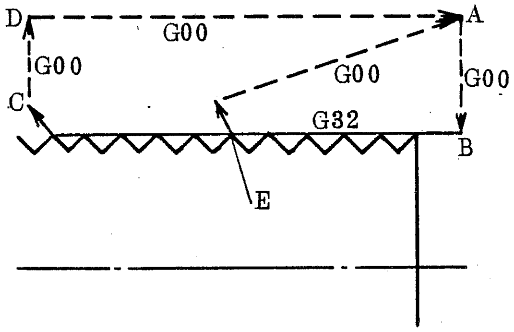
1: spare tools indexed in terms of wear

2: spare tools indexed in terms of time

Spare tools are indexed if they are recorded in the TOOL DATA display when the wear/time TOOL DATA indicate the tool has reached its wear/time life.

Example) Input as illustrated when TNO.1 has reached the end of its life during T0101 machining. It will then continue.

11.6 Temporary Threading Stop



When the FEED HOLD button is pressed at the E-point for example during a threading cycle, a single-step stop is normally performed at the D-point. If parameter $\boxed{\text{U24}}$ is used, however, the same occurs at F-point after the 60° chamfering starting at E-point. If CYCLE START is pressed after single step, X- and Z-axes are returned to the A-point at the same time by rapid feed and thread cutting continue. Restart is made at the Nth time of notching if FEED HOLD is used.

Note 1) During block threading, FEED HOLD remains invalid even if it is pressed near block end. It is also invalid if thread chamfering value is smaller than residual distance.

Note 2) No return is possible to start point if no move command is given three steps before threading block. In this case, therefore, residual steps are run even on stop handling.

```
Example)  M38;
          M03S1000;
          G00X100.Z100.;              F/H ON
          G50X0Z0; <------------ No move command is given
          G92X-100.Z-100.E10.;   to this step.
          M02;
```

11.7 Cautions in EIA/ISO Program

1) Soft limit function only is valid but tool/tail/chuck barrier function is not.

2) VFC does not actuate.

3) Neither graphic-check nor simulation function is provided.

4) The LAYOUT NO. picture does not appear.

5) Single process function cannot be performed. If it is attempted, alarm No."418 "EIA PROGRAM IS DESIGNATED" is displayed.

6) Operation is possible only from memory, not from tape.

7) No MDI function is provided.

Note 1) Parameter setting should not conflict with the life setting, otherwise, life alarm will be given.
Alarm No. 151 is given if wear life has expired when parameter $\boxed{\text{P4}}$=2.

12. TAPE INPUT AND OUTPUT

MAZATROL CAM T-2/T-3 permit the EIA/ISO program paper tape punch data to be stored in memory and in the numerical controller. The following are data transfer methods.

(1) Load

A maximum of 16 programs punched on the paper tape are registered in the NC.

(2) All load

All the paper tape punching programs up to tape end are registered in the numerical control.

(3) Punching

A maximum of 16 programs which have been registered in the NC are punched on the paper tape.

(4) All punching

Paper tape is punched for all the EIA/ISO programs registered in the numerical control.

(5) Compare

A maximum of 16 programs punched on the paper tape are identified with the one with the same work number that is registered in the NC.

(6) All compare

All the paper tape punching programs up to tape end are identified with the ones with same work numbers that are registered in the numerical control.

(7) Rewinding

Rewinding the tape reader

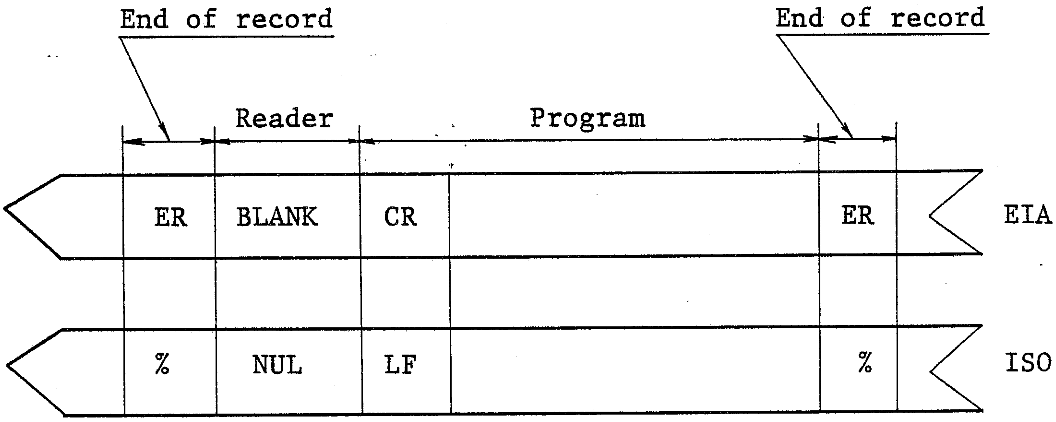
Note 1) Rewinding function is valid only with parameter $\boxed{A16}$. At this time, rewinding is automatically performed after LOAD, ALL-LOAD, COMPARE and ALL-COMPARE.

12.1 Tape Formats

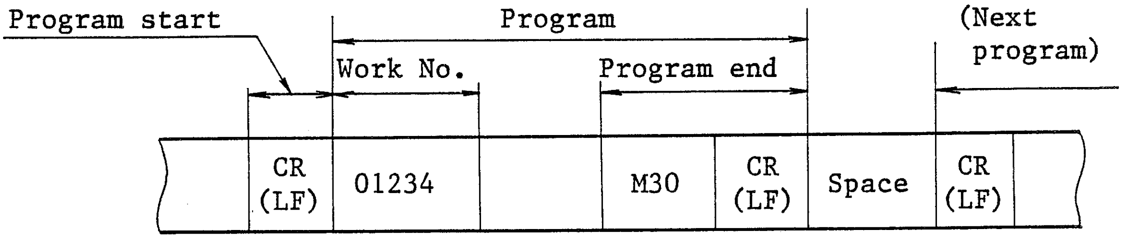
MAZATROL CAM T-2/T-3 uses EIA (RS-244-A) and ISO (R-840) code for tapes.
EIA/ISO change is performed using parameter $\boxed{A20}$. When read, tapes are automatically identified by the initial EOB/LF input.

The following paper tape punch programs are provided.

	EIA	ISO
Reader part	Blank	NUL
Program start	CR or EOB	LF or NL
Program part	-	-
Program end	M02,M30,M99, M198 or M199	M02,M30,M99, M198 or M199
End of record	ER	%



Program



(1) Reader part

The information punched before CR (EIA) or LF (ISO), the first instruction on instruction tape, is called reader part. Whatever other combinations of code punch holes than LF and CR is selectable for the reader part since their parity check is not carried out.

(2) Program start

The CR (EIA) or LF (ISO) punched immediately after the reader part signifies the start of the program.

(3) Program part

The punch part holes between the program start and end are named program part. The program part consists of one or more programs where movement and other machining instruction informations are punched.

Work number holes are punched at the head of the each program part in accordance with address O. Work numbers are registered in the numerical control but not displayed on programs when paper tape is read. The programs without work number punch holes can be read, provided their work number is registered in advance.

(4) Program end

When any of codes M02, M30, M99, M198 and M199 is read, the blocks up to corresponding one are registered in the memory as one program. Address-O can also cause program end if no M code is given.

M-codes between control-out "(" and control-in ")" are not regarded as program end commands.

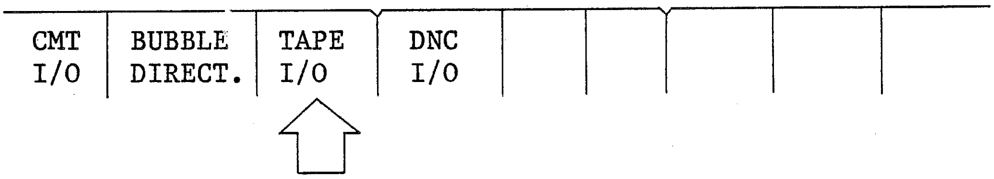
(5) End of record

The end-of-record punches at the both ends of tapes:

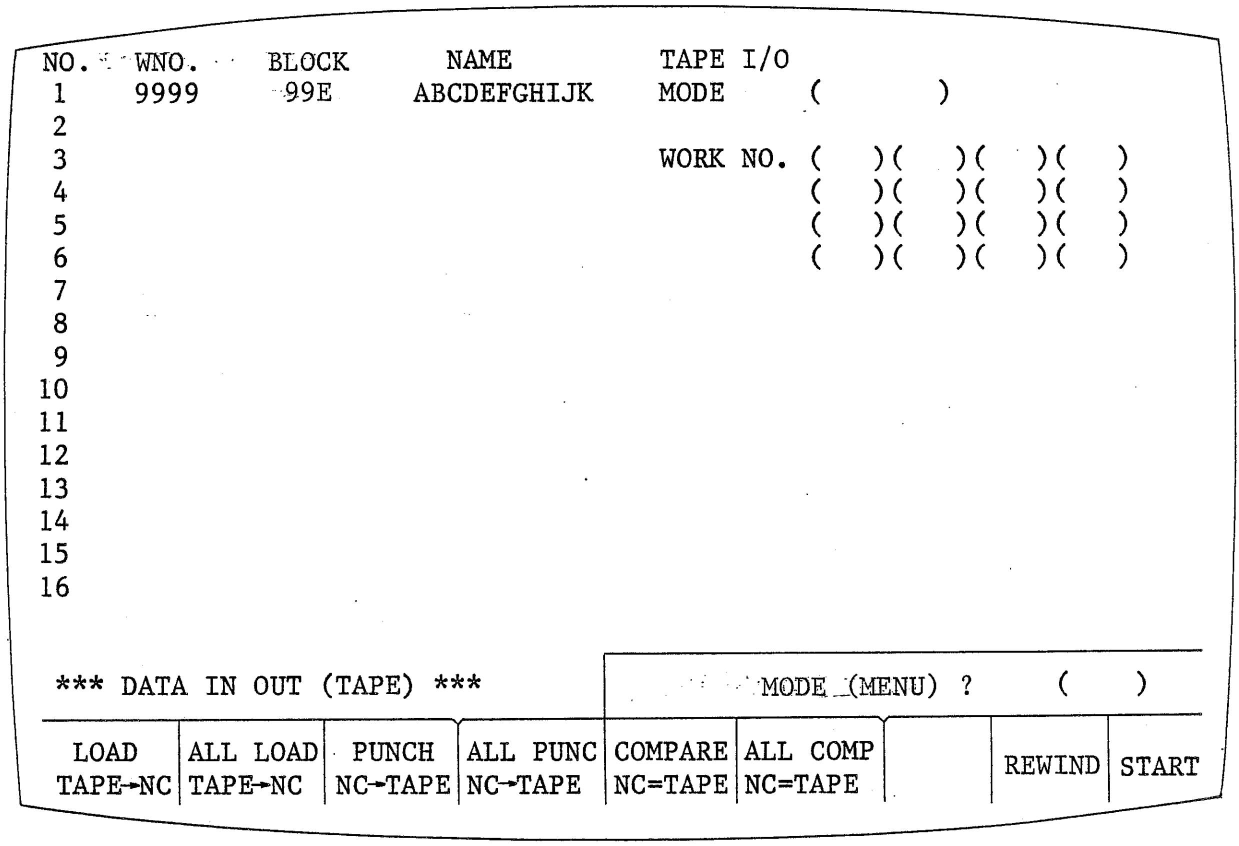
- o Stop rewinding (winder is provided.)
- o Complete the tape read on ALL LOAD and ALL COMPARE.

12.2 Call of Displays

Call the following menu by pressing ▣ , PROGRAM, PROGRAM FILE and DATA IN OUT menu keys in order.



Then, call the following display by pressing the TAPE I/O menu key.



```
NO.  WNO.    BLOCK      NAME         TAPE I/O
 1   9999     99E     ABCDEFGHIJK    MODE      (        )
 2
 3                                   WORK NO.  (    )(    )(    )(    )
 4                                             (    )(    )(    )(    )
 5                                             (    )(    )(    )(    )
 6                                             (    )(    )(    )(    )
 7
 8
 9
10
11
12
13
14
15
16

 *** DATA IN OUT (TAPE) ***                 MODE (MENU) ?     (    )
```

Tape I/O display

12.3 Handling

i) LOAD

(1) Put the paper tape into the reader.

(2) Press LOAD . I/O mode (LOAD) will be displayed.

(3) Select work numbers.
Up to 16 work numbers can be selected. They are handled according to the paper-tape punch program format.

(i) No work numbers are given to the paper tape program.
The selected work number program is read.

(ii) Work numbers are entered in the paper tape program and compared with selected work numbers. If they differ, an alarm is given. Otherwise, the corresponding work number program is read. Only when work numbers are input to the paper tape program, is LOAD possible without selecting them. At this time, only the paper punch work numbers for the first program read out of tape, are read.

(4) Press START .
Menu display inverts and paper tape is read.

(5) Menu display returns to former position to indicate completion of transfer.
Tape is rewound if rewind function is valid.

Note 1) No LOAD is possible when a program with the same number is stored in the numerical control memory. ERASE or PROGRAM RENUMBER of the program from the numerical control is performed.

Note 2) Loading can be interrupted by pressing the RESET key. In this case the data up to interrupt point are recorded.

ii) ALL LOAD

(1) Put paper tape into the reader.

(2) Press ALL LOAD . I/O mode (ALL LOAD) will appear.

(3) Press START .
Menu display inverts and all the programs are stored in the numerical control memory.

(4) Menu display returns to former position to indicate the completion of transfer.
Tape is rewound if rewinding function is valid.

Note 1) Programs are recorded in the numerical controller when paper tape punch work numbers are input. No loading is possible without these work numbers.

Note 2) No loading is possible for programs with same work numbers which are already in the numerical controller. ERASE or RENUMBER of programs in the numerical controller must be performed.

Note 3) Loading can be interrupted by pressing the RESET key. In this case the programs up to interrupt point are recorded.

iii) PUNCH

(1) Put the paper tape into the punch.

(2) Press PUNCH . I/O mode (PUNCH) appears.

(3) Select work numbers.
Up to 16 work numbers are selectable. Those not recorded in the numerical controller can not be.
No MAZATROL program punch is possible.

(4) Press START .
Menu display inverts and the program punch is performed.

(5) Menu display returns to its former position to indicate the completion of punching.

Note 1) Loading can be interrupted by pressing the RESET key. In this case punch tape format is not correct.

iv) ALL PUNCH

(1) Put the paper tape into the punch.

(2) Press ALL PUNCH . I/O mode (All PUNCH) appears.

(3) Press START .
Menu display inverts and the program punching is performed for all EIA/ISO programs stored in the numerical control.

(4) Menu display returns to its former position to indicate completion of ALL PUNCH.

Note 1) Programs are punched in the order in which they are recorded in the numerical control.

Note 2) Bubble memory (option) specification - all the EIA/ISO programs registered on bubble memory can be entered by punching.

Note 3) Loading can be interrupted by pressing the RESET key. In this case, the punch tape format is not correct.

v) COMPARE

(1) Put the paper tape into the reader.

(2) Press COMPARE . I/O mode (COMPARE) appears.

(3) Select work numbers.
Use the same work-number selection method as i) LOAD. However the work numbers not in the numerical control memory are not selected.

(4) Press START .
Menu display inverts and the paper tape is read. The programs are compared with those stored in numerical control memory. If they do not agree, reading will stop and an alarm message be displayed.

(5) Menu display returns to its former position to indicate completion of the transfer.

Note 1) Transfer can be interrupted by pressing the RESET key during COMPARE.

Note 2) No correct processing is performed in some cases where the programs without an end (M02, M30, M99, M198, M199) are compared by setting up the parameter for address-0 program end.

vi) ALL COMPARE

(1) Put the paper tape into the reader.

(2) Press ALL COMPARE . I/O mode (ALL COMPARE) appears.

(3) Press START .
Menu display inverts and all the programs up to the end of record are subject to v) COMPARE.

(4) Menu display returns to its former position to indicate completion of the transfer.
Tape is rewound if rewinding function is valid after normal completion of the transfer.

Note 1) The programs with paper tape punch work numbers are compared with those with same work numbers stored in the numerical control memory. The programs on the tape without work numbers are not compared.

Note 2) Transfer can be interrupted by pressing the RESET key in COMPARE.

Note 3) No correct processing is performed in some cases where the programs without an end (M02, M30, M99, M198, M199) are compared by the parameter set up for address-0 program end.

vii) 1) REWIND

(1) Put paper tape into the reader with the rewind mechanism.

(2) Press REWIND . I/O mode (REWIND) appears.

(3) Press START .
Menu display inverts and the tape is rewound.

(4) Menu display returns to its former position to indicate completion of processing.

Note 1) This operation is possible only when the rewinding function is valid.

Note 2) Tape is not read. RESET key can not be used to stop.

12.4 Parity H/V

MAZATROL CAM T-2/T-3 has parity check functions PARITY H/V for checking whether tape is correct to prevent tape punch errors. There are two kinds of parity H and V.

(1) Parity H

Parity H error occurs when the code holes read in any part except the reader part of paper tape are as follows.

① EIA codes: even number of holes
② ISO codes: odd number of holes

(2) Parity V error occurs when odd number of codes are input to a block between CR(LF) and CR(LF) in any part except the reader part of paper tape.

This function's validity can be determined by using parameter $\boxed{A24}$.

If the former is the case, a space is inserted before CR(LF) when odd number of codes are given to one block by tape punch.

13. CODES

The M-Code Table is included in the operating manual.

14. PARAMETERS

Tabulated below are only the parameters relating to the EIA/ISO program. See the operating manual for further details.

Addresses	Names
P4	Spare tool indexing
P6	Inch/metric changeover
P8	Initial setting when power on G01/G00
P9	Initial setting when power on G98/G99
P11	G53 MAZATROL coordinate system select valid/invalid
U4	Fixed point return position X-axis
U5	Fixed point return position Z-axis
U17	EIA/ISO restart, single step stop
U18	Rapid feed dry run valid/invalid
U19	Threading or tapping dry run valid/invalid
U24	Feed hold during threading valid/invalid
U27	Pattern of the escape after G71 or G72 turning cycle
U28	Measuring tolerance range 1
U29	Measuring tolerance range 2
U33	Threading finish allowance
U36	Thread chamfer value
U43	Grooving relief amount
U50	Drill relief amount
U62	Minimum threading feed
U70	Coefficient of increasing feedrate when $\boxed{U27}$ =2
U73	Value for automatically setting work coordinate system on the return to EIA reference point (X-axis)
U74	Value for automatically setting work coordinate system on the return to EIA reference point (Z-axis)

Addresses	Names
U76	Initializing value for display (X-axis)
U77	Initializing value for display (Z-axis)
A5	EIA cycle starting conditions
A6	Offset-commanded axis move by EIA
A15	RS232C parameter for serial tape reader/puncher
A16	RS232C parameter for serial tape reader/puncher
A18	RS232C baud rate for serial tape reader/puncher
A19	Tape reader rewind code
A20	N/C tape output code
A21	Tape punch output-number of space codes
A22	Tape punch output-number of space codes between programs
A23	Tape punch output-number of head and tail feed
A24	T/V check - valid/invalid
A29	Detection of the tape reader program end
A30	Serial/parallel change-over
A31	#-code for EIA punch
A32	:-code for EIA punch
S15	F data for restart/TPS