FANUC SYSTEM 3T-MODEL D

OPERATOR'S MANUAL

CONTENTS

B-53504E/02

I. GENERAL

1. GENERAL

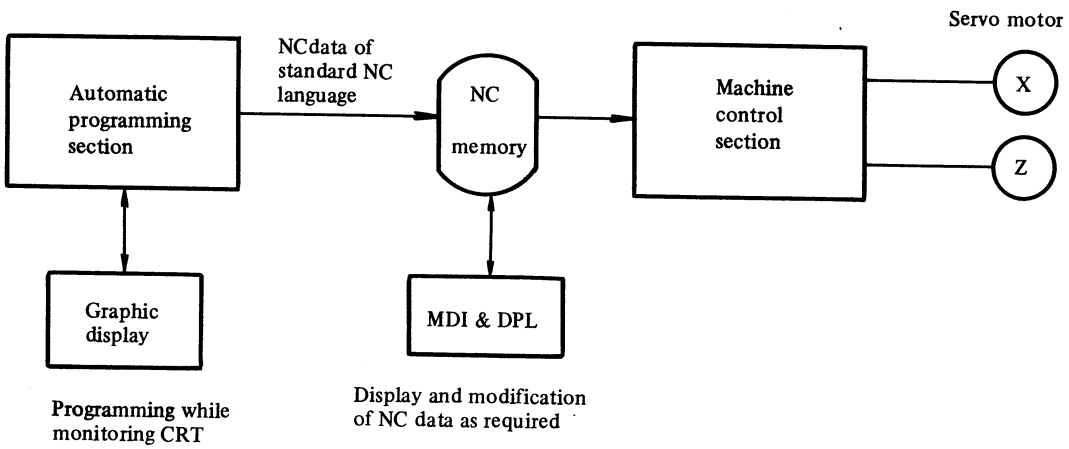
The FANUC SYSTEM 3T-MODEL D is an SNC recently developed by FANUC. SNC means an NC in which functions of an epoche-making automatic programming system "Symbolic FAPT" are assembled into CNC, and this "Symbolic FAPT" allows even unexperienced NC programmers to produce NC data for machining at once simply by depressing buttons according to the instructions on the graphic display CRT. Accordingly, the SNC is a controller which can execute both programming and machining concurrently at a site. The SNC can execute machining according to a program soon after programming it at a site, or it can execute another programming during machining.

The Symbolic FAPT produces NC data of the standard NC language (EIA or ISO format), and loads them into built-in memory of SNC (NC memory). Machining is made according to the NC data in memory (also called NC program).

During machining, another program can also be produced by using Symbolic FAPT function. When the NC memory has been unloaded after machining, another program being prepared at the automatic programming section is transferred to the NC memory.

Since NC data loaded in NC memory are prepared in standard NC language, an operator who knows the standard NC language can directly check or modify NC data of standard NC language in NC memory by using the MDI & DPL unit of SNC.

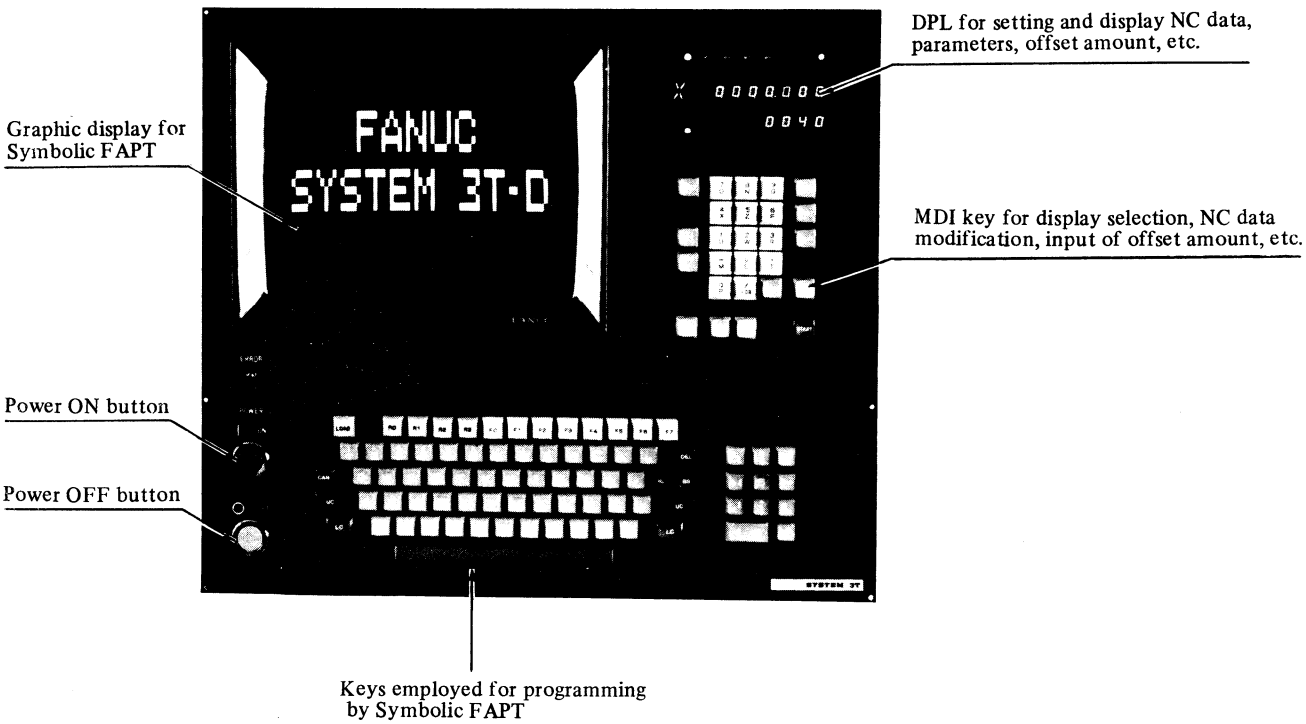
(Since NC data can be transferred again to NC memory after modifying NC data at the automatic programming section without any need of modifying NC memory data, it is not necessary to modify NC memory data usually.)



Chapter II describes the programming and operation by Symbolic FAPT, while chapter III describes the transfer of NC program produced by Symbolic FAPT to NC memory.

After understanding these description, you can start machining at once, referring to the machine tool builder's operation manual.

2. DESCRIPTION OF FRONT PANEL



The operator's panel is clearly divided into two sections; one is used for Symbolic FAPT, while the other is used for machining according to NC data, so that another program is producible during machining, as an excellent feature of FANUC SNC.

The left graphic display and lower keys are used for Symbolic FAPT, while the LED display at the right upper part is used for machining according to NC data.

3. FANUC SYSTEM 3T-MODEL D

① The FANUC SYSTEM 3T-MODEL D consists of the machine control unit (FS3T-A) and automatic programming unit (P-D with Symbolic FAPT) which are directly connected to each other.

② The operation and functions of the machine control unit are the same as those in former FANUC SYSTEM 3T-MODEL A.
Machine tools are fully controlled by this control unit.

③ The automatic programming unit (P-D with Symbolic FAPT) is exclusively provided to automatically produce NC machining information.

④ The NC data thus produced automatically by Symbolic FAPT are not directly used for machining, but they are once transferred to the machining memory on the machine control unit (FS3T-A) side. The subsequent operation belongs entirely to the FS3T-A operation, irrespective of the Symbolic FAPT and CRT screen.

⑤ A CRT is mounted on the P-D side. Accordingly, it cannot apply to various display of the machine control unit (FS3T-A), but it can be used for automatic programming of Symbolic FAPT only.

⑥ Neither various pieces of information on NC machine tool nor various pieces of information on FS3T-A can be stored on the P-D side, but the control information of machine tools is processed on the FS3T-A side and displayed on the display unit of FS3T-A only.

⑦ The machining information can be input to FS3T-D by the following two methods.

(1) Method of inputting the machining information into the machining memory of the machine control unit by MDI operation.
Since this operation is the same as in NC unit for ordinary MDI input, you are requested to be familiar with NC tape format, etc.

(2) Method of transferring automatic programming results into the machining memory of the machine control unit (FS3T-A) after automatic programming has been made on the P-D side by using the Symbolic FAPT.
This operation is made, while monitoring the CRT screen.

⑧ The NC machining information data automatically produced by the Symbolic FAPT can also be loaded and stored into memory cassette via memory cassette adapter instead of transferring them into the machining memory on the machine control unit (FS3T-A) side.
The following information can be loaded into the memory cassette.

(1) Input/output from Symbolic FAPT side
- Family program
- Material file
- Tooling file
- NC machining data (This data can be output from Symbolic FAPT to bubble cassette only. It cannot be input from bubble cassette to Symbolic FAPT)

(2) Input/output from machine control unit (FS3T-A) side
- NC machining data

⑨ When the FS3T-D power source is turned off, the memory contents on the Symbolic FAPT side (P-D) are deleted.

4. NOTES ON READING THIS MANUAL

4.1 The function of an NC machine tool system does not depend only on the NC, but on the combination of the machine tool, its magnetic cabinet, servo system, the NC, operator's panels, etc. It is too difficult to describe the function, programming, and operation in various combinations of them. This manual generally describes them from the standpoint of the NC. So, for a particular NC machine tool, refer to the manual issued by the machine tool builder, which should take precedence over this manual.

4.2 This manual addresses as many subjects as possible. But it would become too voluminous to write out what should not and cannot be done, which are very numerous. So the functions which are not referred to as possible in it should be interpreted as "impossible".

4.3 Notes refer to detailed and specific items. So, when a note will be encountered, terms used in it sometimes will not have been explained it. In such a case, first skip the note, then return to it after having read the manual in outline for details.

4.4 The memory cassette and the memory cassette adapter in this manual is a bubble cassette and a bubble cassette adapter, respectively.

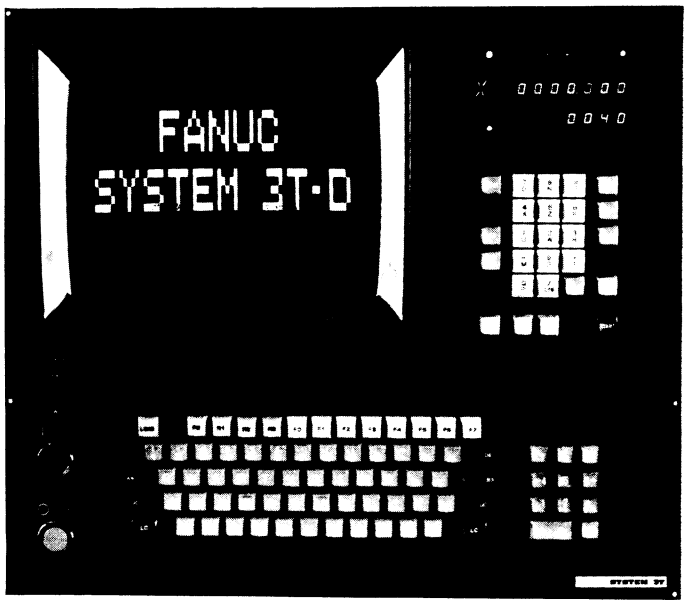
II. PROGRAMMING AND OPERATION USING Symbolic FAPT

1. Symbolic FAPT IS

The Symbolic FAPT is an epoch-making NC automatic programming system which enables even unexperienced NC operators to prepare NC command data immediately, simply by observing instructions on the graphic display CRT. Thus, it is no longer necessary to keep promises on NC tape in mind nor remember NC automatic programming language.

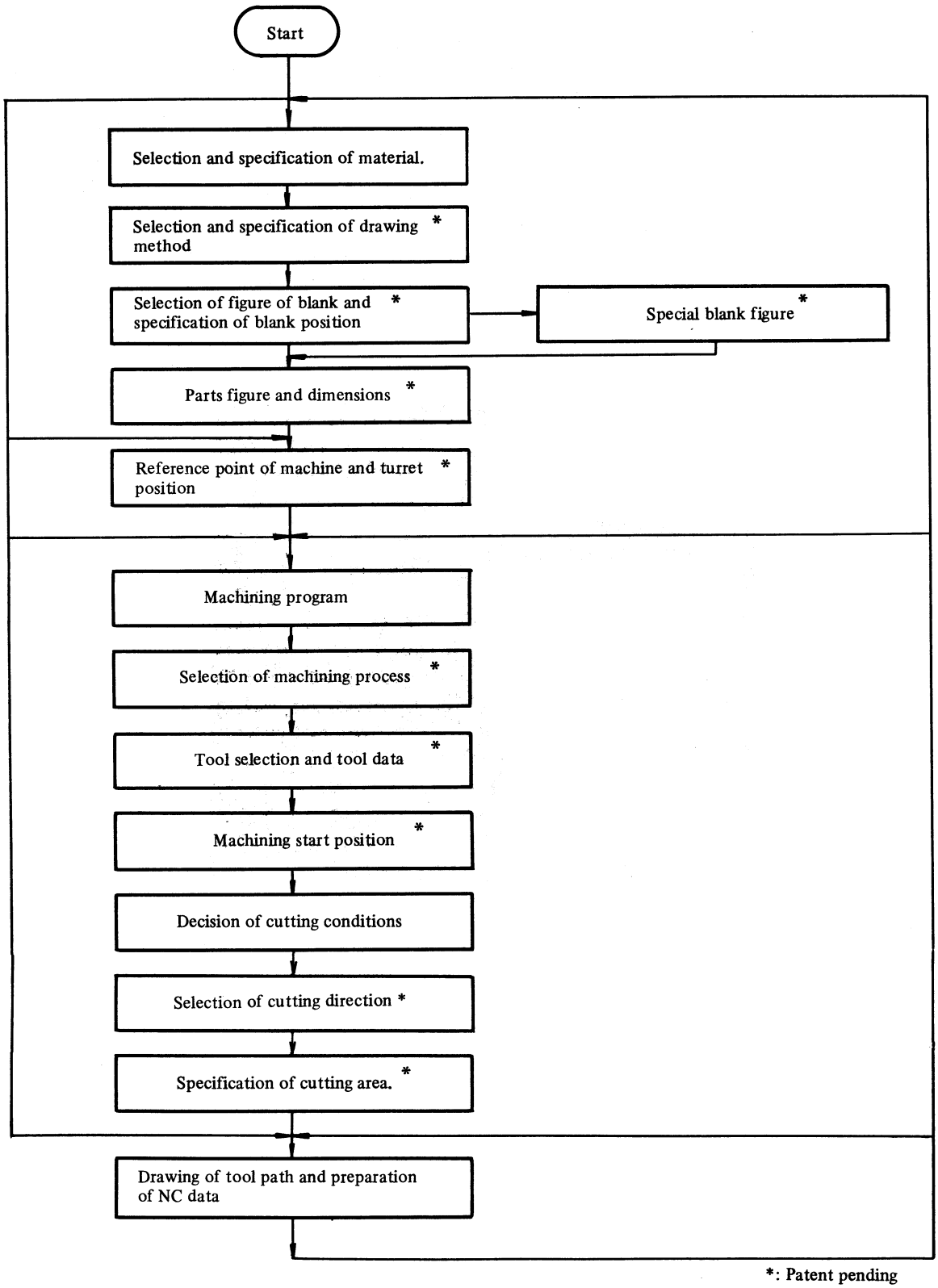
Each process from design drawings to NC command data preparation is entirely advanced by the conversational operation with the graphic display CRT. For ① setting of coordinate system, ②selection of figures of blank and input of dimensions, ③ input of parts figures and dimensions, ④ selection of machining process and decision of cutting conditions, etc., the operator has only to answer questions from the CRT.

The parts figure can be inputted simply by depressing corresponding symbol keys on the console panel according to the profile of a work described on design drawings. When dimensions and various data are inputted, various pieces of reference information are illustrated on the CRT from time to time, and questions are given to the operator using daily language. When data are inputted, the figure of blank and parts profile are drawn immediately, and automatic calculations of NC command data are started. The tool path is concurrently displayed as figures.



Symbolic FAPT FLOW

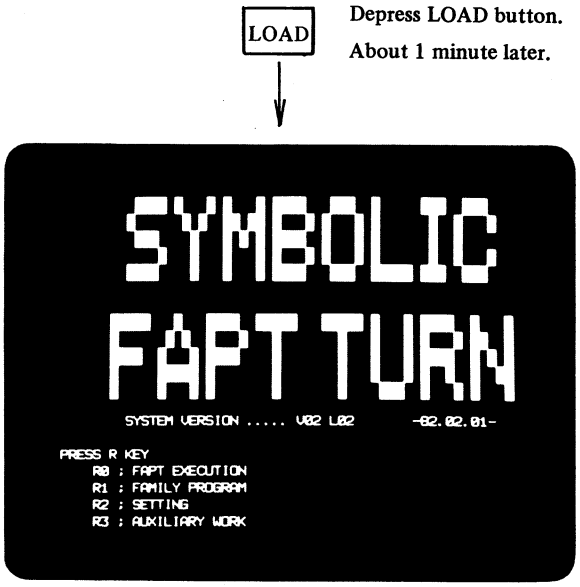
The preparation procedure of NC data by using Symbolic FAPT function is illustrated as follows.



*: Patent pending

2. LOADING OF Symbolic FAPT

At first, load the system program of Symbolic FAPT by the following way.



Various jobs are selectable by depressing R keys, while the above pattern appears on CRT screen.

R0 ..... **FAPT EXECUTION**

Depress R0 key for NC programming, while monitoring the CRT screen.

R1 ..... **FAMILY PROGRAM**

For calling a family program from memory cassette, depress R1 key.

For using the family program, the memory cassette and memory cassette adapter are necessary.

R2 ..... **SETTING**

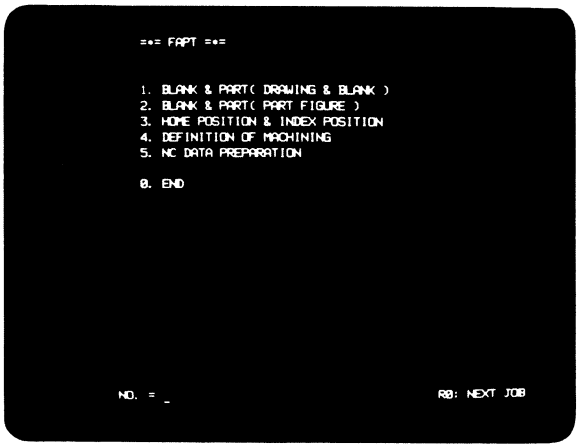
R2 key is used for input/output of "system parameters", "material files", and "tooling files". For storage of material files and tooling files, the FANUC cassette and the FANUC cassette adapter are necessary.

R3 ..... **AUXILIARY WORK**

R3 key is used for setting "machine tool file (MTF)".

(Note 1) It is not necessary for operator and part programmer to operate R2 (setting) and R3 (auxiliary work) keys.

Depress [R0] key ..... FAPT EXECUTION



By depressing the [R0] key when the initial pattern appears on CRT screen, processing proceeds to **"FAPT EXECUTION"**, and the above picture is displayed.

The CRT screen shows the execution procedure menu, and menu number are inputtable optionally. The execution is started with the programming process having the input menu number. For example, to select menu number 1, depress [1] [SP] . If [SP] key only is depressed, it selects a flickering menu number.

However, the process of NC data preparation cannot be selected without inputting the parts figure.

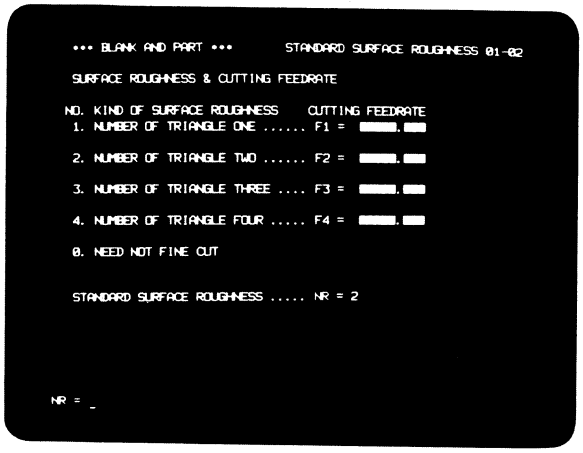
3. EXECUTION OF Symbolic FAPT

The programming can be performed, while monitoring the CRT screen.

3.1 Blank and Part (Drawing and Blank)

(Setting of standard surface roughness)

☛ [SP] **or** [1] [SP] **..... Blank & part (Drawing & blank)**



A menu is displayed for selecting the surface roughness.
Select the menu according to the number of ▽ marks attached to machining drawings.
Observe the following operation, if almost parts of a machining drawing are marked with ▽▽ .

[2] [SP] **..... Selection of ▽▽**

When the following display appears at the lower part of CRT screen, depress [R0], [R1], or [R3] key.

***** PRESS R KEY *** R0: NEXT PAGE R1: COLLECTION R3: KILL**

[R0] . . . . . Processing proceeds to the next step.

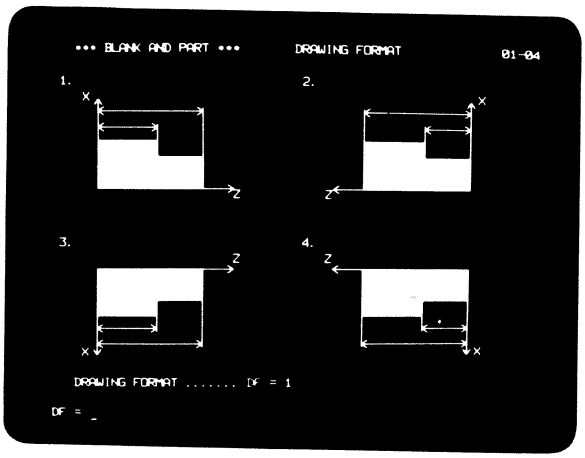
[R1] . . . . . Depress this key when you desire to modify input contents being displayed on CRT screen.

[R3] . . . . . Depress this key when you desire to abort the FAPT execution halfway.

After you have been familiar with this operation, you can automatically forward processing without depressing [R0] key each time. For automatic advance, you can also set the time of changing the CRT screen in the unit of seconds.

☛ [R0] **..... Proceed to the next processing. (Not required, if automatic processing is set, in advance).**

Selection of drawing format



If dimension values are described on drawings based on the left side, select menu number [1] or [3], and if they are described based on the right side, select menu number [2] or [4] so as to facilitate dimensional data input during programming.

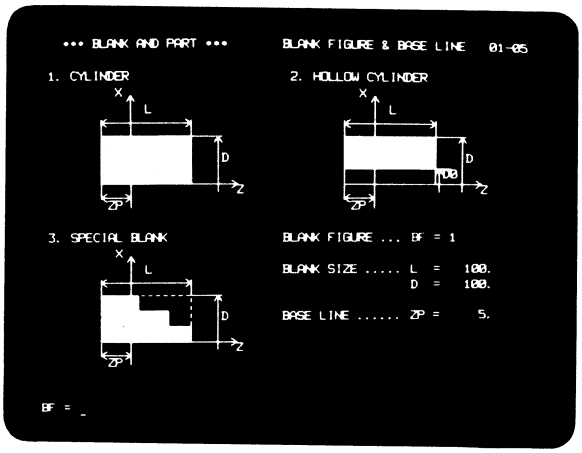
(Note 2) In order to easily understand the relation between positive direction and negative direction of respective setting angles, and the other variables when inputting the tool figures, setting angles, etc., it is recommended to select menu number [3] or [4] when the tool post is mounted on the front side and also select menu number [1] or [2] when the tool post is mounted on the rear side

Operate the keyboard as shown below, for example.

[1] [SP] ..... **Selection of drawing format**

[R0] ..... **Proceed to the next step (Not required, if automatic processing is set, in advance)**

Blank figure



A picture to specify the blank figure is displayed on the CRT screen.

- **BLANK FIGURE . . . . . BF**
  A round rod, a hollow rod, or a special figure blank is selected by the menu numbers 1, 2, 3.
- **BLANK SIZE . . . . . L, D, D0**
  The system asks you the length (L) and diameter (D) of blank. It also asks you the inner diameter (D0) in case of a hollow rod.
- **BASE LINE . . . . . ZP**
  This designates where does the coordinate system to be programmed exist in the blank.
- **THICKNESS . . . . . TX, TZ**
  If a special figure is selected as a blank figure, the system asks you the X component and Z component of the surplus thickness.

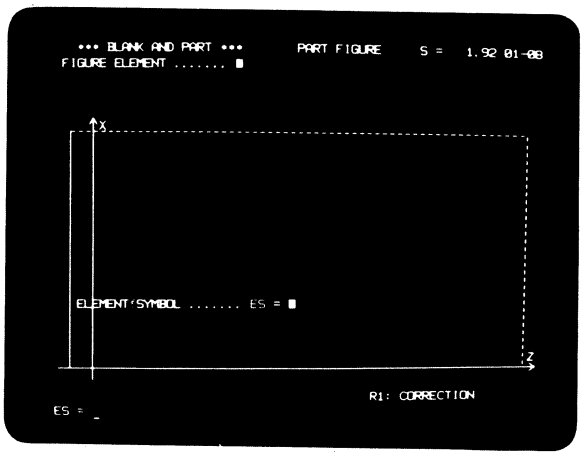
Input data when the system asks you questions.

R0 **..... Proceed to the next step.**
(Not required, if automatic processing is set, in advance.

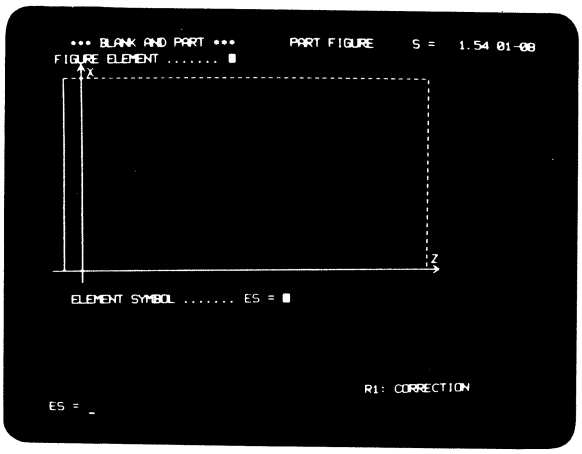
3.2 Blank and Part (Part Figure)

3.2.1 Plotting of program coordinate axis and blank figure

The coordinate axes conforming to the previously selected drawing format and blank figure are plotted on the CRT screen.



F1 key is turned on.



F1 key is turned off.

- The blank is plotted by a dotted line.
- The left end of the blank is plotted by a solid line to indicate the chuck side.
- When F1 key is turned on;

  The blank is plotted over the entire CRT screen.

  The conversation message overlaps the blank.
- When F1 key is turned off;

  The blank is plotted to such a size as it does not overlap the conversation message on the CRT screen.
- The blank is automatically scaled so that it is accomodated with the above range, irrespective of its dimensions.

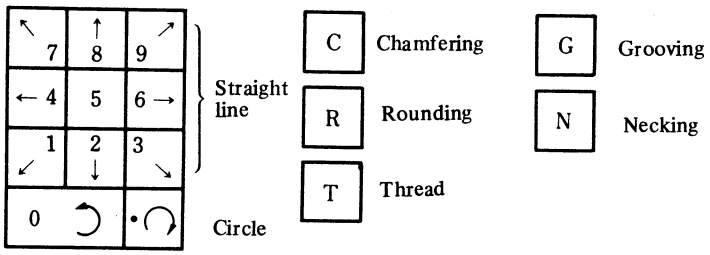
- If the blank is long laterally, the conversational area is provided on the lower side of the CRT screen, and if it is long longitudionally, the conversational area is provided on the right side of the CRT screen. This decision is automatically done according to the dimensions of input blanks.

The system requests you to input the part figure by the following display in the conversational area on the CRT screen.

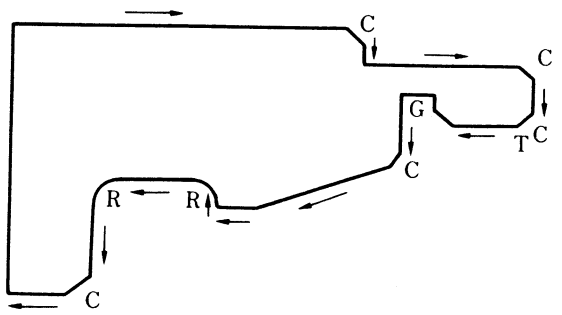
ELEMENT SYMBOL ..... ES = ■

3.2.2 Part figure input

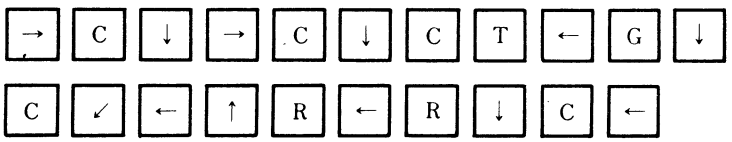
The part figure can be input by depressing the figure symbol keys sequentially along the profile of a part. part. Ten keys for numerical input and alphabetic keys indicating the chamfering, rounding, threading, grooving, and necking are employed as figure symbol keys.



(Example)



For a figure illustrated above, input the following keys along the drawing.



The system will ask you necessary dimension, each time these figure symbol keys are depressed.

Surface roughness

The system asks you the "surface roughness", each time the figure element symbol like a circle, a straight line, etc. is input. The already designated "standard surface roughness" is automatically displayed on the CRT screen at this time.

If this value is allowable, depress [SP] key only.

If you desire to change the surface roughness at that place only, select the number of ▽ marks 1, 2, 3, or 4.

Start point

The system asks you the start point at the first step of figure input only.

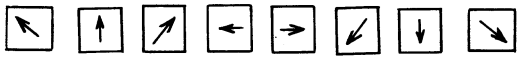
SPX . . . . . Diameter value at start point

SZ . . . . . Z value at start point

PE . . . . . Whether a part exists on the left side or on right side along a series of a figure profile to be input, is selected by 1 or 0.

These questions are made for the first figure element only, when a part figure is input

Straight line



(1) **ELEMENT SYMBOL ..... ES = ■**

Input an arrow along the profile figure when the system asks you the above question.

For example, input [↖][SP].

(2) **SURFACE ROUGHNESS ..... SR = ■**

Input the number of ▽ marks indicating the surface roughness when the system asks you the surface roughness.

(3) The system will ask you the following items sequentially, starting with the top item by displaying these items on the CRT screen.

END POINT	**DX =**	Diameter value at end point
	Z =	Z value at end point
TANGENT LAST	**TL =**	Whether the profile touches the last figure or not.
TANGENT NEXT	**TN =**	Whether the profile touches the next figure or not.
ANGLE FROM Z	**A =**	Angle from Z axis

You have to answer only the dimensions described on the drawing. Depress [SP] key only, if a dimension is not found on the drawing.

The system does not always ask all the above items, depending upon the input arrow directions, but it asks you necessary numerical values only, and you have to input only those described on the drawing.

(4) If data are insufficient, **"KEY IN AGAIN"** is displayed.

Circle

(1) **ELEMENT SYMBOL ..... ES =**

Input either key or key along the progressive direction of the circle when the system asks you the above question.

(2) **SURFACE ROUGHNESS ..... SR =**

Input the number of ▽ marks indicating the surface roughness when the system asks you the above question.

(3)

END POINT . . . . . DX =		Diameter value at end point
	Z =	Z value at end point
TANGENT LAST . . . . . TL =		Whether the profile touches the last figure or not.
TANGENT NEXT . . . . . TN =		Whether the profile touches the next figure or not.
RADIUS . . . . . R =		Radius value of circle.
CENTER . . . . . CDX =		X value at circle center (diameter coordinate value)
	CZ =	Z value at circle center

The system asks you the above questions sequentially by displaying these items on the CRT screen. Answer only such dimensions as are known from the drawing. Depress SP key only, if a corresponding dimension is not found on the drawing.

(4) If data are insufficient or if an invalid numerical value was inputted, KEY IN AGAIN is displayed on the CRT screen.

(5) R key may be used for inputting a circle. For details of R key, refer to the item of rounding.

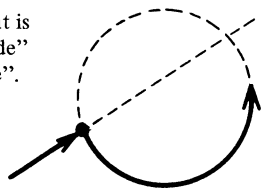
(6) The system asks you the following question when a profile figure transfers from a straight line to circular arc or from a circular arc to a circular arc.

SELECT START POINT (INTERSECTION OF LINE-CIRCLE)
(→ : RIGHT ← : LEFT ↑ : ABOVE ↓ : BELOW)

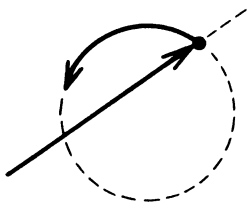
The parts figure input comprises two methods when a profile figure transfers from a straight line or a circular arc to the circular arc.

When keys were depressed;

The start point is either "left side" or "lower side".

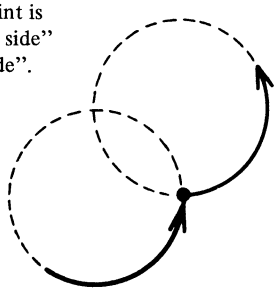


The start point is either "right side" or "upper side".

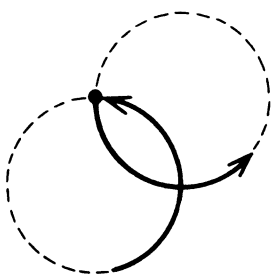


When keys were depressed;

The start point is either "right side" or "lower side".



The start point is either "left side" or "upper side".



When the system asks you **"SELECT START POINT"** to identify the following two possible methods from each other, you have to select the right side (→), left side (←), above (↑) or below (↓) by the [→] [←] [↑] or [↓] key.

Chamfering [C]

Chamfering is designated by [C], which indicates the capital of chamfer.

(1) **ELEMENT SYMBOL ..... ES =■**

Answer [C] when the system asks you the above question.

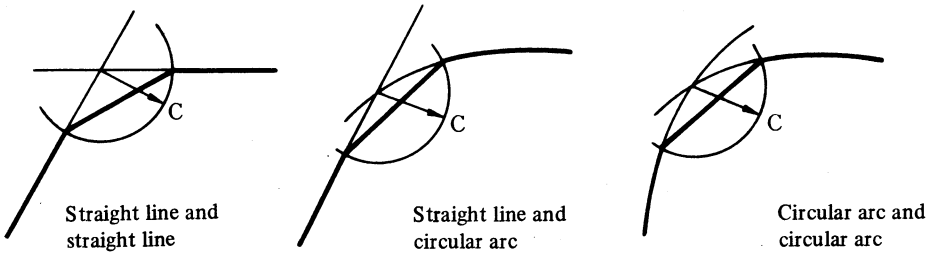
(2) **SURFACE ROUGHNESS ..... SR =■**

Input the number of ▽ marks indicating the surface roughness when the system asks you the above question.

(3) **CHAMFER ..... C =■**

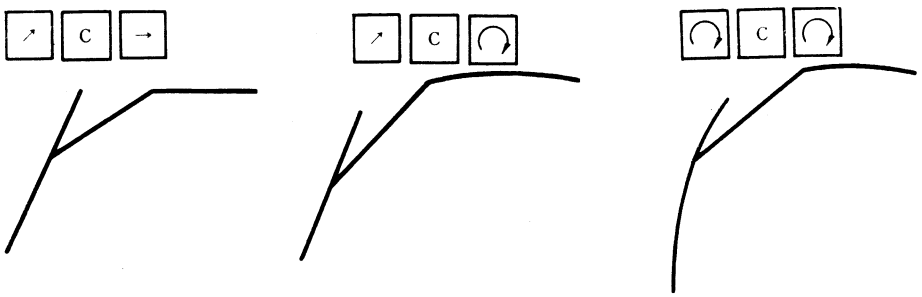
Input the chamfering amount for the above question.

(4) The chamfering amount is figured as shown below.



Straight line and straight line | Straight line and circular arc | Circular arc and circular arc

(5) The chamfering is displayed on CRT screen as shown below when a figure is input.



Rounding [R]

Rounding is designated by [R], which indicates the capital of rounding.

(1) **ELEMENT SYMBOL ..... ES =■**

Answer [R] when the system asks you the above question.

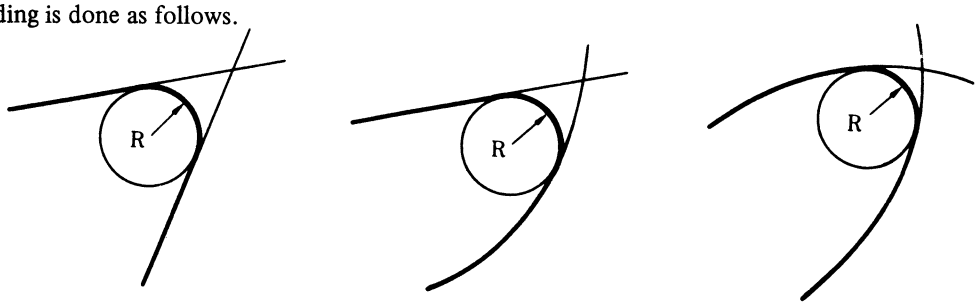
(2) **SURFACE ROUGHNESS ..... SR =■**

Input the number of ▽ marks indicating the surface roughness when the system asks you the above question.

(3) **RADIUS ..... R =■**

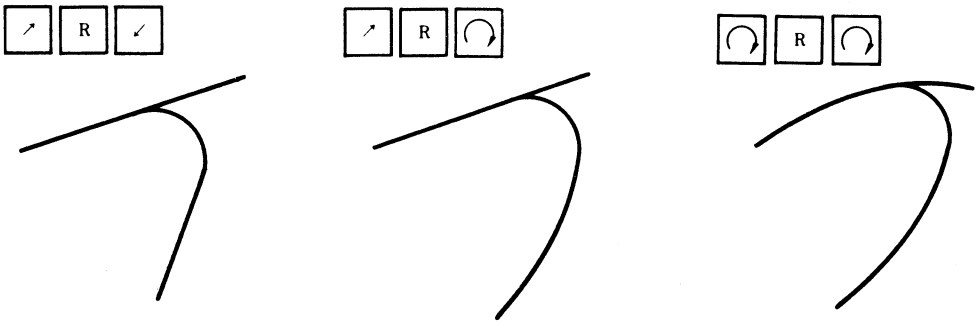
Input the radius value for rounding when the system asks you the above question.

(4) Rounding is done as follows.



Straight line and straight line | Straight line and circular arc | Circular arc and circular arc

(5) Rounding at this figure input time is displayed as a figure on the CRT screen as shown below.



(Note 1) Keys [↷] , [↶] *and* [R] *should be selectively used for specifying a circular arc as described below.*

- *(1) A tangential circle being caught by 2 straight lines, (2) a tangential circle being caught by a straight line and a circular arc, and (3) a tangential circle being caught by circular arcs can be specified by depressing* [R] *key as a figure element key, if their radiuses are know, as illustrated above.*
- *Circular arc can also be specified by depressing* [↷] *or* [↶] *key.*
- *If the coordinate values of a circular arc center, a start point and an end point are written on a drawing, the rounding can be specified by using* [↷] *or* [↶] *key.*
- *For such a circular arc as it touches the last and next figures and its radius only are known, depressing* [R] *key is recommended for the convenience.*

(Threading) [T]

Threading is designated by [T], which indicates the capital of thread.

(1) **ELEMENT SYMBOL ..... ES =** ■

Answer [T] when the system asks you the above question.

(2) **ON WHICH ELEMENT? ..... EE =** On which element is threading done?
LENGTH ..... LT = Length of thread
LEAD ..... LD = Lead of thread
MULTIPLE ..... NT = Number of threads of screw
(0: LAST 1: NEXT)

The system asks you these questions, starting with the top item by displaying them on CRT screen.

(3) **"ON NEXT ELEMENT" or "ON LAST ELEMENT"?**

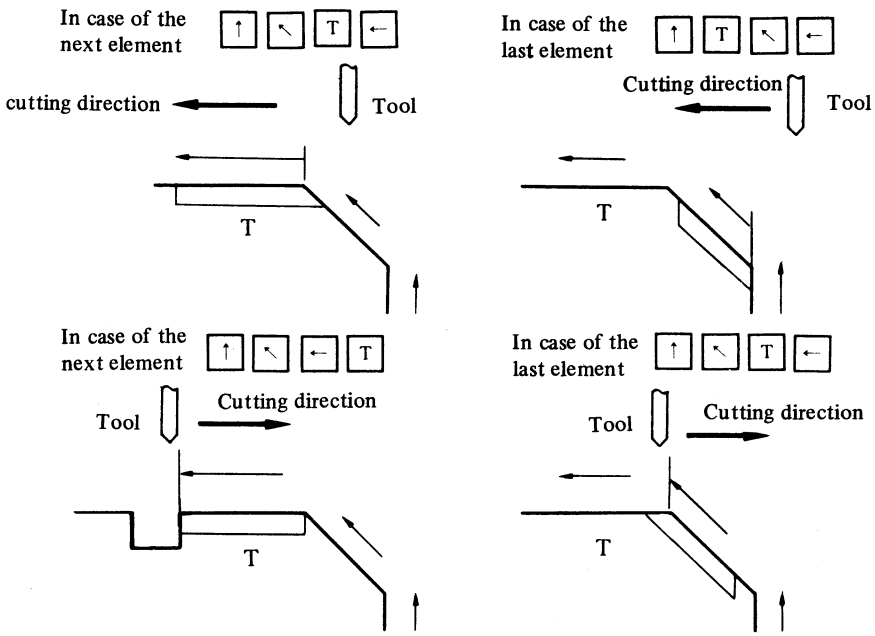
[T] is inputted before and after arrows, in general.

It indicates whether threading is made on a figure element to be input or on the last figure element already input.

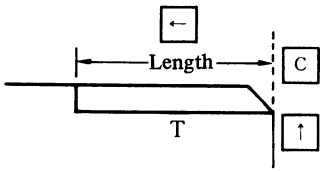
"ON NEXT ELEMENT" ..... Input "1".

"ON LAST ELEMENT" ..... Input "0".

Identify **"ON NEXT ELEMENT"** and **"ON LAST ELEMENT"** from each other in the following illustration.



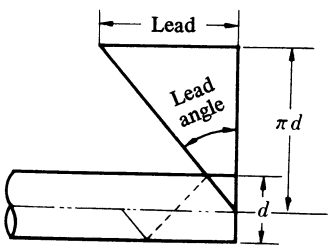
(Note 2) When chamfering was made by [C], the length of thread can be calculated from the intersection of linear element before chamfering.



(4) Lead length, pitch, and number of thread

The lead is defined as the Z-axis direction length when a screw returns to the original angle after turning once. In case of a one-thread screw, the pitch coincides with the lead. In case of a three-thread screw, 1/3 of lead corresponds to the pitch.

The lead of a taper screw is also calcurated in the Z-axis direction.



(Grooving) [G]

Grooving is designated by [G], which indicates the capital of groove.

(1) **ELEMENT SYMBOL ..... ES = ■**

Answer [G] when the system asks you the above question.

(2) **SURFACE ROUGHNESS ..... SR = ■**

Input the number of ▽ marks indicating the surface roughness when the system asks you the above question.

(3) **ON WHICH ELEMENT? ..... EE =** — On which element is grooving made?

DIRECTION .....DN= — Grooving direction

WIDTH .....WT= — Grooving width

DEPTH .....DT = — Grooving depth

R/C OF CORNER (ALONG ELEMENT)

1'ST ..... KC = CV =

2'ND ..... KC = CV =

3'RD ..... KC = CV =

4'TH ..... KC = CV =

} Designation of groove shape

The system asks you these questions sequentially, starting with the top question by displaying them on the CRT screen. Input a numerical value for each question.

(4) **"ON NEXT ELEMENT" or "ON LAST ELEMENT"?**

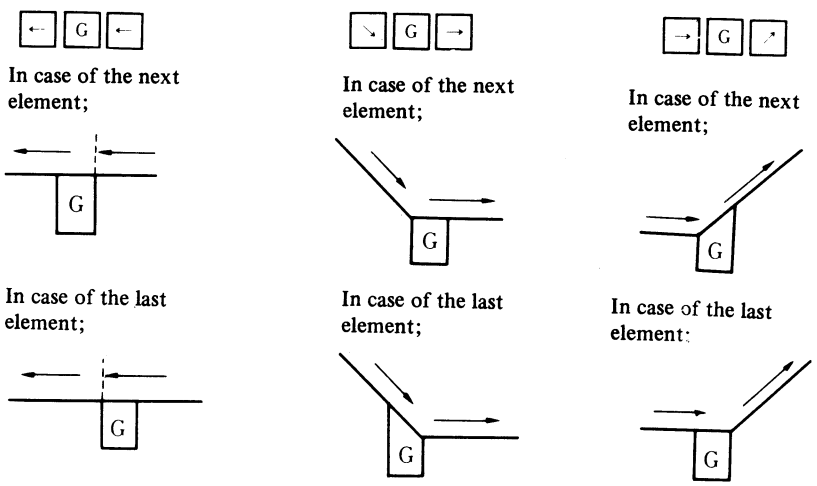
[G] is input before and after arrows, in general.

It indicates whether gooving is made on a figure element to be input or on the last figure element already input.

"ON NEXT ELEMENT" ..... Input "1".

"ON LAST ELEMENT" ..... Input "0".

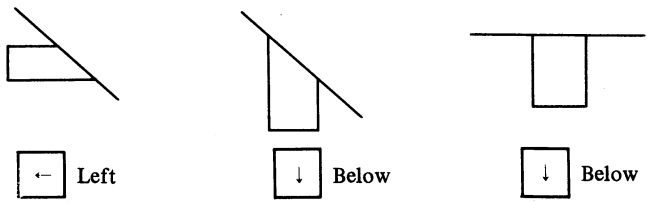
Identify **"ON NEXT ELEMENT"** and **"ON LAST ELEMENT"** from each other in the following illustration.



(5) **DIRECTION ..... DN =**

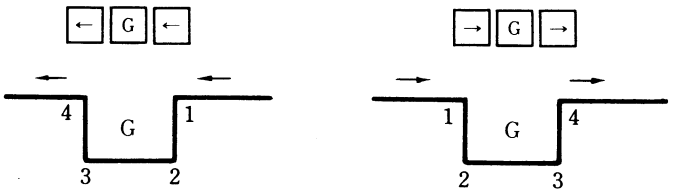
(→ : RIGHT ←: LEFT ↑ : ABOVE ↓ : BELOW)

Designate the grooving direction by the arrow such as right (→), left (←), above (↑) or below (↓) when the system asks you the above question.



(6) After inputting the width and depth, the system asks you the figure at groove corners. Numbers 1 to 4 are allocated to the corners of each groove in the sequence of figure input.

(Example)



Designate whether the figures of respective corners 1 to 4 are "rounded (R)", "chamfered (C) or "left unmachined" according to the following questions being displayed on the CRT screen.

R/C OF CORNER (ALONG ELEMENT)

1'ST ..... KC =	**CV =**	Figure at the 1'st corner
2'ND..... KC =	**CV =**	Figure at the 2'nd corner
3'RD..... KC =	**CV =**	Figure at the 3'rd corner
4'TH ..... KC =	**CV =**	Figure at the 4'th corner

- For rounding, depress [R] for **"KC="** question, and input a radius value for **"CV="** question.
- For chamfering, depress [C] for **"KC="** question, and input a chamfering amount for **"CV="** question.
- If a corner is neither rounded or chamfered, and it is left undone as it is, depress [SP] key only, or depress [R0] key to proceed to the next CRT screen when the system asks you the question.

Necking [N]

Necking is designated by [N], which indicates the capital of necking.

(1) **ELEMENT SYMBOL ..... ES = ■**

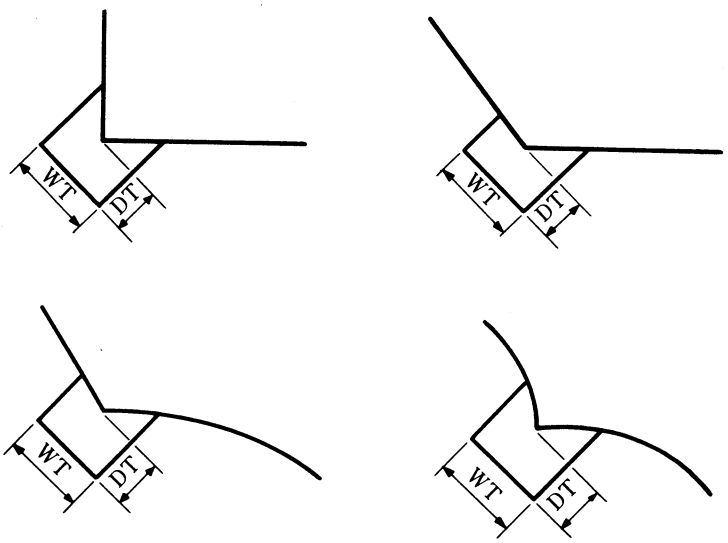
Answer [N] when the system asks you the above question.

(2) **SURFACE ROUGHNESS ..... SR = ■**

Input the number of ▽ marks indicating the surface roughness when the system asks you the above question.

(3) **WIDTH** ..... **WT** = Necking width
DEPTH ..... **DT** = Necking depth

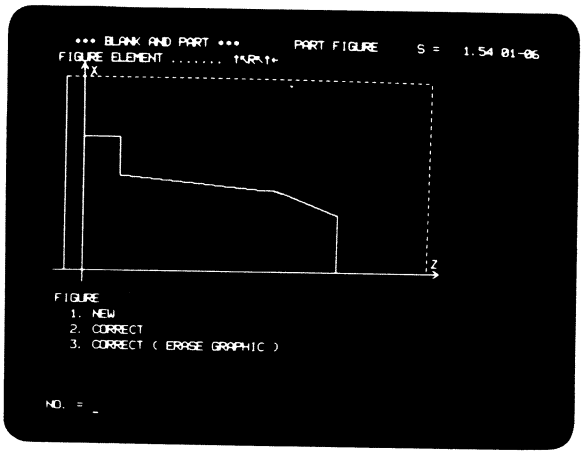
Input the necking width and depth for the above questions. The necking direction is always 45° with reference to Z-axis. The necking width and depth are as illustrated below.



End of figure input

After all parts figure have been input, depress [SP] key only, when the system asks you a element symbol.

ELEMENT SYMBOL ..... **ES** = ■



Menu

The parts figure already input by that time is drawn again on the CRT screen, and the following menu is displayed at the lower part of the CRT screen.

FIGURE

1. **NEW**
2. **CORRECT**
3. **CORRECT (ERASE GRAPHIC)**

Compare the figure on the drawing with the figure on the CRT screen, and check the latter for a possible input failure. Perform one of the following procedures according to the present circumstances.

(1) Select menu number [1], if you want to input the figure again from the beginning after erasing the parts figure on the CRT screen.

(2) Select menu number [2], if you want to modify the parts figure without erasing the parts figure being displayed on the CRT screen.

(3) Select menu number [3], if you want to modify the figure, while sequentially displaying the previously inputted parts figure step by step, each time the [SP] or [R0] key is depressed after erasing the parts figure now being displayed on the CRT screen once.

(4) Depress either [SP] or [R0] key, if you want to proceed to the next process without any modification when the system asks you with the display and menu as shown above.
A picture for selection of execution sequence is displayed, and the machine home position and index position menu number 3 will flicker. The flickering menu number is selected by depressing [SP] key only.

(5) The contents which are previously programmed by another programmer may appear on the CRT screen. In such a case, the programmed parts figure is erased by selecting [1] for new programming.

3.2.3 Input and modification of part figure data

(Arithmetic operation and function calculation)

The arithmetic operation (addition, subtraction, multiplication, division) and optional function calculations can also be executed when inputting dimensions.

Sym-bol	Arithmetic contents	Examples	Remarks
[+]	Addition	15 [+] 27.75 [+] 4.8	
[−]	Subtraction	178.5 [−] 25.25	
[*]	Multiplication	57.5 [*] 81.3	* indicates X.
[/]	Division	128 [/] 3.5	/ indicates ÷.
[T]	tangent	[T] 23 [T] [(] 23 [)]	The unit of angle is degree. Other arithmetic symbols are employable inside parentheses.
[S]	sine	[S] 15 [S] [(] 15 [)]	"
[C]	cosine	[C] 35 [C] [(] 35 [)]	"
[A] [S]	arc sine	[A] [S] 0.25 [A] [S] [(] 0.25 [)]	"
[A] [C]	arc cosine	[A] [C] 0.37 [A] [C] [(] 0.37 [)]	"
[A] [T]	arc tangent	[A] [T] [(] 3, 5 [)]	Parentheses are required without fail. [A] [T] (numerator and denominator).
[R]	$\sqrt{}$	[R] 177 [R] [(] 177 [)]	Optional arithmetic expressions are employable inside parentheses, if any.
[P]	power	[P] [(] 1.5, 4 [)] [P] [(] 3, 2.5 [)]	These examples mean 1.5^4 and $3^{2.5}$, respectively.

- Parentheses [(] and [)] are employable quadruply or less.
- Arithmetic symbols are employable whenever a dimension is asked. They are employable by an optional number, until one line is fully filled.
- An arithmetic expression may be used conveniently when a cutting condition is changed, for example. For setting the feedrate to 75%, multiply the displayed numerical value by 0.75.
- It is recommended for using Symbolic FAPT to avoid mental calculation as much as possible, but use this arithmetic function.

Input of incremental value

If the system asks you a dimension and you want to input an incremental value from the present position, press [I] key after an incremental value as shown below. [I] indicates the capital of incremental.

(Example) 50.45 [I]

With [I] specification, the data in X-axis direction is radius value.

Change of figure element symbols

Depress [R1] key when the system asks you the following question.

ELEMENT SYMBOL ..... ES = ■

The question advances or retreats according to whether [F0] key is being turned off or on at that time.

[R1] with [F0] turned off ..... Question advances
[R1] with [F0] turned on ..... Question retreats

The question place can be confirmed by a flickering figure element symbol at the upper part of the CRT screen.

(Insertion example)

When depressing [R] with [↗] [↷] [→] [↷] ..... [↗] [↷] [R] [→] [↷]

↑ Flickering

When depressing [C] with [↗] [G] [→] [↓] ..... [↗] [G] [→] [C] [↓]

↑ Flickering

(Deletion example) Depress [DEL] key

When depressing [DEL] with [↗] [↷] [→] [↷] ..... [↗] [→] [↷]

↑ Flickering

When depressing [DEL] with [↗] [G] [→] [↓] ..... [G] [→] [↓]

↑ Flickering

(Exchange)

For exchanging figure symbols, (1) delete old one after inserting new one, or (2) insert new one after deleting old one.

- When [R1] is kept depressing in case of retreating a question, the question returns to the first figure element symbol. This state remains unchanged even when depressing the [R1] key any times.
- When [R1] is kept depressing in case of advancing, ■ flickers. This state remains unchanged, even if [R1] is depressed longer.

Change of numerical value

The system will ask you necessary numerical values after figure element symbols have been input. If a numerical value was input by mistake, a correct numerical value can be input again after selecting the question place optionally by depressing [R1] key.

The question place advances, each time [R1] key is depressed, if [F0] key is turned off. The question place retreats, each time [R1] key is depressed, if [F0] key is turned on.

Select an optional question, and input a correct numerical value again.

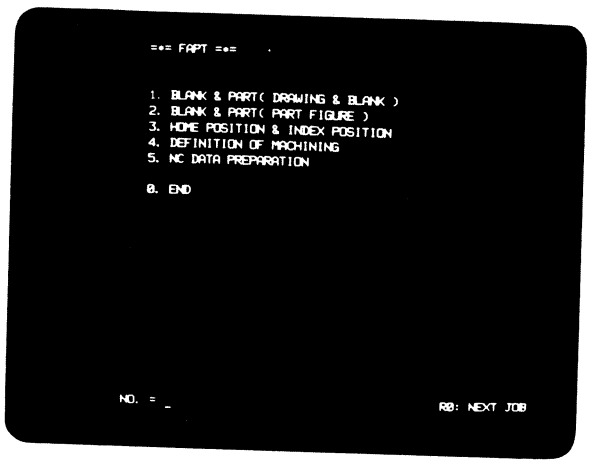
[R1] with [F0] key turned c^ff ..... Question advances

[R1] with [F0] key turned on ..... Question reteats

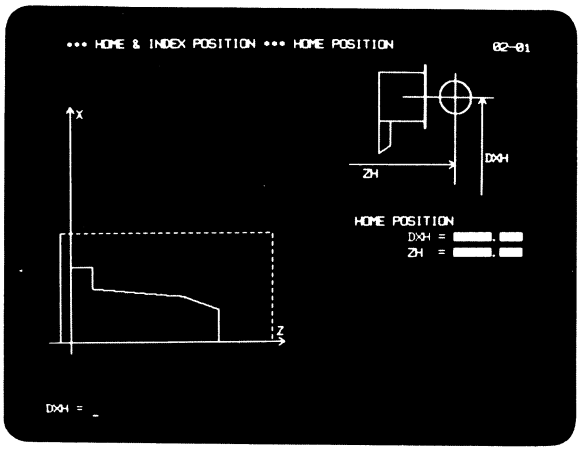
(Note 1) If a figure element of data was corrected, execute its subsequent processing by depressing either [SP] key or [R0] key without fail. If processing is not executed after correction, its results may differ from expected, because subsequent data are newly produced only by executing processing again using the corrected figure element or data.

3.3 Home and Index Position

If a menu number [3] is selected when the CRT screen displays the menu for selection of execution sequence as shown below, the system asks you "HOME POSITION".



Setting of home position



Question

The system asks you the following question as displayed in the above CRT screen.

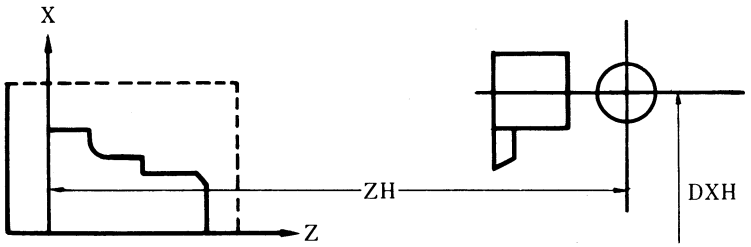
HOME POSITION	Home position
DXH =	Diameter value
ZH =	Z value

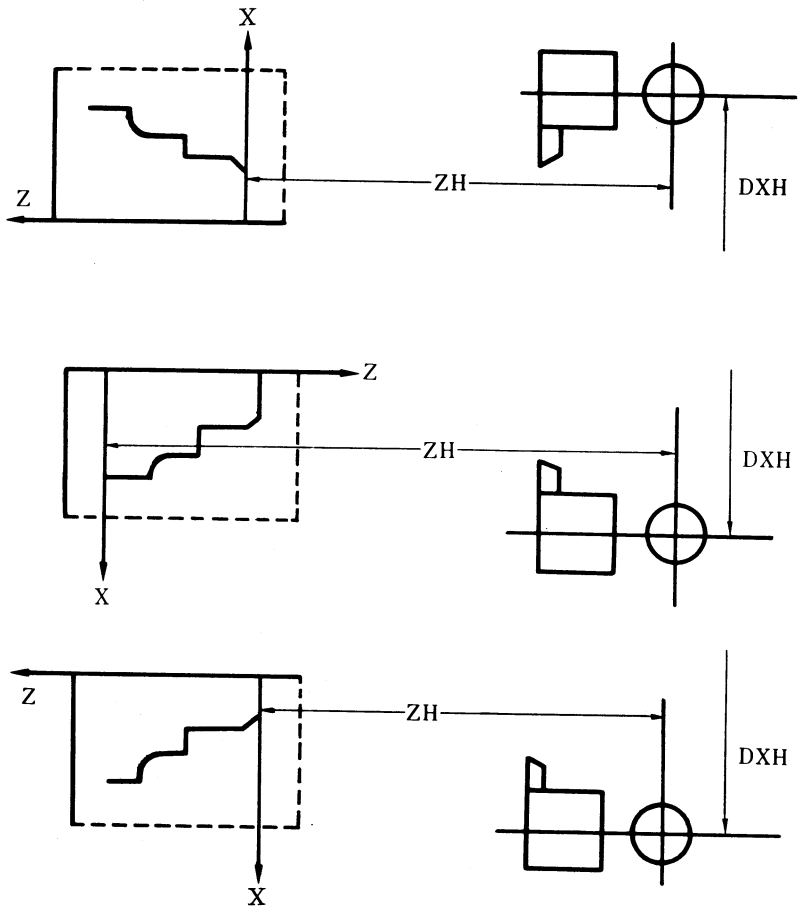
Input the home position of your NC lathe.

If data are preset and set values now being displayed on the CRT screen are allowable without any need of their change, depress [SP] or [R0] key, and the next picture is displayed on the CRT screen.

(Note 1) Input the distances from the programmed coordinate axis as the coordinate values showing the home position.

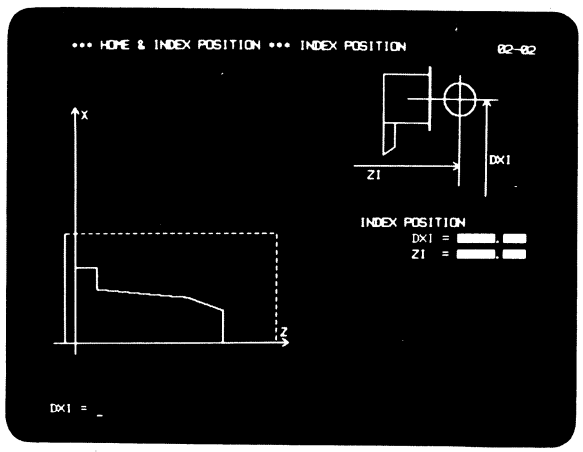
The signs of home position is Z-axis direction (HZ) should be positive, irrespective of the Z-axis direction of the programmed coordinate axis. The machine home position is used for indicating the relation between the program coordinate system and the machine coordinate system.





The above cautions consern the home position. They also apply to the index position mentioned at the next item, correspondingly.

Setting of index position



} Question

The system asks you the following question as displayed in the above CRT screen.

INDEX POSITION	Index position
DXI =	Diameter value
ZI =	Z value

Input the coordinate values at the index position of turret for machining the programmed part in response to the question from the system.

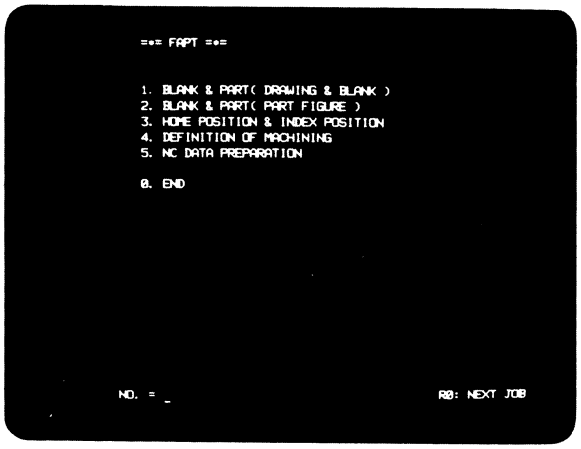
Depress either [SP] or [R0] key, if you want to use set data as it is.

3.4 Machining Definition

3.4.1 Kind of process

Depress [R0] key after inputting the index position (Depressing [R0] key is not required, if the system has been preset to the automatic processing mode). Now, the following menu for selection of execution sequence will appear on the CRT screen.

[R0] (Not required, if automatic processing is set in advance.)

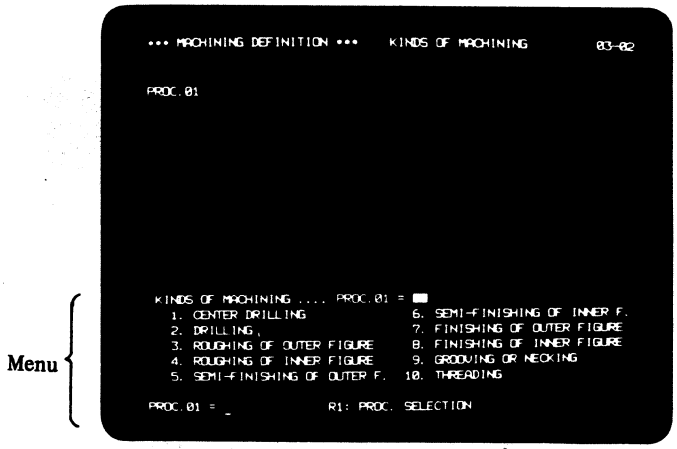


Menu number [4] flickers on the CRT screen. When [SP] key only is depressed, the flickering menu number is selected, and processing will shift to the execution process of **"DEFINITION OF MACHINING"**.

You can optionally select other menu numbers or change the execution process for the purpose of checking or modifying input data, while the adove menu is being displayed on the CRT screen.

[4] [SP] ([SP] key only, if [4] is flickering)

By the above operation, the CRT screen will change to the menu which defines how to machine the programmed parts figure.



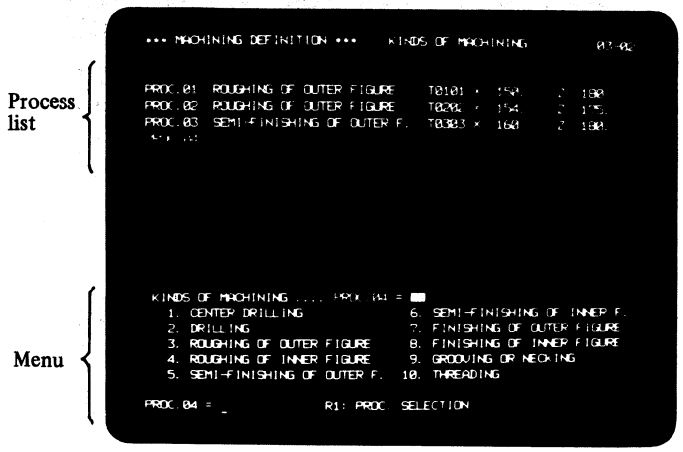
The kinds of machining required for lathing will be displayed as a menu at the lower part of the CRT screen. The menu contents are as shown below.

1. **CENTER DRILLING**
2. **DRILLING**
3. **ROUGHING OF OUTER FIGURE**
4. **ROUGHING OF INNER FIGURE**
5. **SEMI-FINISHING OF OUTER F.**
6. **SEMI-FINISHING OF INNER F.**
7. **FINISHING OF OUTER FIGURE**
8. **FINISHING OF INNER FIGURE**
9. **GROOVING OR NECKING**
10. **THREADING**

The system asks you the machining method of process number 1 by displaying **PROC. 01 =** ■■ on the CRT screen. If you answer the question by depressing [3] and [SP] keys, it means that **"ROUGHING OF OUTER FIGURE"** has been selected.

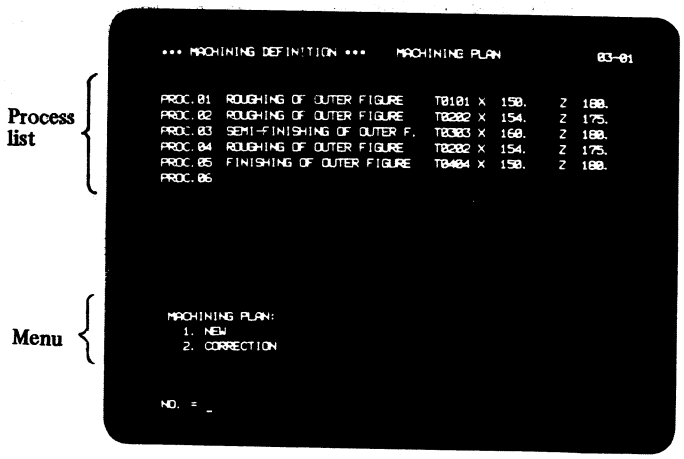
3.4.2 Specification of multiprocess

If a process is newly added when several processes are being displayed on the CRT screen, the specified process is added last.



If [7] and [SP] keys are depressed when the above display appears on the CRT screen, **"FINISHING OF OUTER FIGURE"** is added to process number 04.

3.4.3 Modification and change of process.



Assume the following menu is displayed at the lower part of the CRT screen.

MACHINING PLANE:

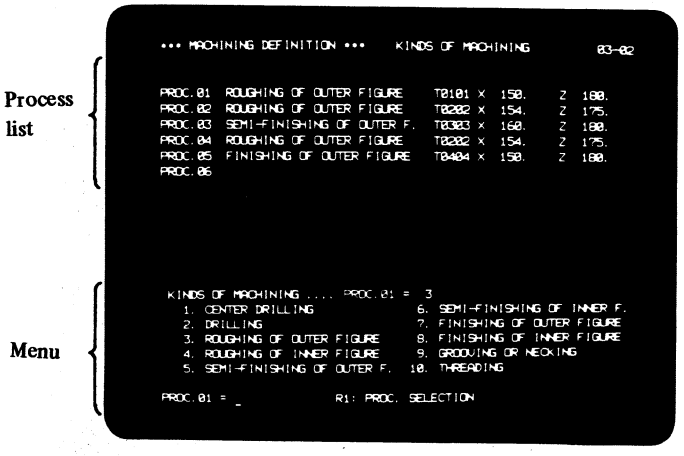
1. NEW	New definition
2. CORRECTION	Correction

When [1] and [SP] keys are depressed, all previous processes are cancelled and processes are newly specified from the beginning.

When [2] and [SP] keys are depressed, a part of processes is corrected or changed.

In practical machining, NC machining data are automatically created through all processes according to the sequence of the process list being displayed on the CRT screen.

[2] [SP] (For correction)



The process number in the process list changes, each time [R1] key is depressed when the above display appears on the CRT screen.

Select an optional process number by depressing the [R1] key, and correct the machining data.

(Example of insertion)

PROC. 01 DRILLING
PROC. 02 ROUGHING OF OUTER FIGURE
PROC. 03 FINISHING OF OUTER FIGURE
PROC. 04

New machining data will be loaded in process number 4 normally in the above example.

① Select a modification place by depressing [R1] key. (Select PROC. 03, for example)

② Input desired machining data to be inserted by menu number. (Depress [4] and [SP] keys, for example.)

Display will change as follows:

PROC. 01 DRILLING
PROC. 02 ROUGHING OF OUTER FIGURE
PROC. 03 ROUGHING OF INNER FIGURE ← Inserted.
PROC. 04 FINISHING OF OUTER FIGURE
PROC. 05

(Example of deletion)

① Select the process number to be deleted by depressing [R1] key.

② Depress [DEL] and [SP] keys.

(Exchange)

① Delete old machining data after inserting new one, or

② Insert new machining data after deleting old one.

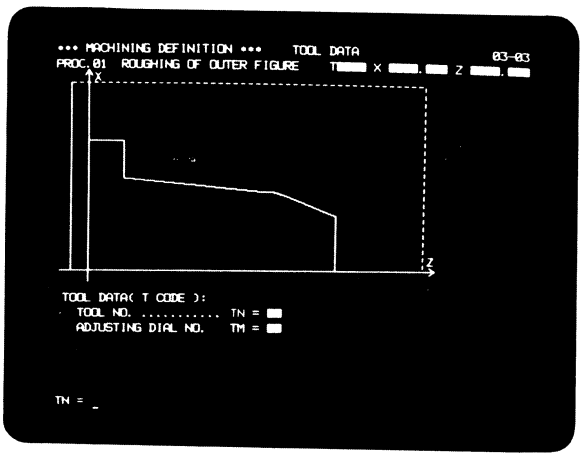
(Note 1) Whether the process shifts backward or forward when [R1] key was depressed, is determined according to whether [F0] key is turned on or off, respectively.

(Note 2) Whenever new machining data are loaded, you should specify data on necessary machining methods (cutting area, tool data and others) without fail.

3.4.4 Setting of data for machining

(Tool data (T code))

Each time a machining process is loaded and displayed on the process list of CRT screen, the display changes, and the system asks you data required for the definition of machining.



Input tool number and tool offset number in response to the following system question.

TOOL DATA (T CODE):

TOOL NO. ..... TN = ■■

ADJUSTING DIAL NO. ..... TM= ■■

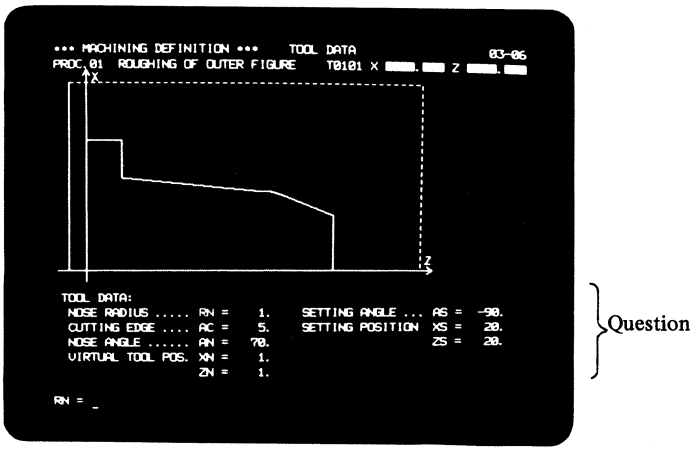
For example,

TOOL NO. ..... TN = [1] [SP]

ADJUSTING DIAL NO. ..... TM = [1] [SP]

Input data are converted into T code, and displayed at the right upper part of the CRT screen. A question on tool figure data will be displayed at the lower part of the CRT screen.

Tool figure data and tool setting method



The system asks you the tool figure data as shown below.

(For lathe)

TOOL DATA

NOSE RADIUS	**RN** =	Nose radius
CUTTING EDGE.....	**AC** =	Cutting edge
NOSE ANGLE	**AN** =	Nose angle
VIRTUAL TOOL POS....	**XN** =	Virtual tool position
	ZN =	Virtual tool position
NOSE WIDTH	**WN** =	Nose width
SETTING ANGLE	**AS** =	Setting angle
SETTING POSITION	**XS** =	Setting position
	ZS =	Setting position

(For drill)

TOOL DATA

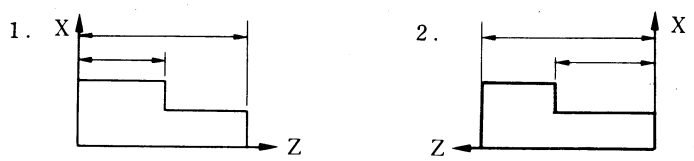
NOSE ANGLE	**AN** =	Nose angle
DRILL DIAMETER.....	**DD** =	Drill diameter
SETTING POSITION ...	**XS** =	Setting position
	ZS =	Setting position

The Symbolic FAPT performs interference check and offset calculation according to tool figure. Be careful since tool figure data will unfavorably affect machining results, if they are not input accurately.

Set the tool figure data and setting method according to the following figure.

- When the tool post is set at the rear side.

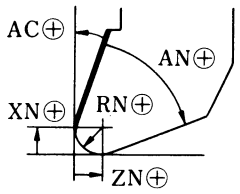
  In this case, select [1] or [2] as the drawing format in order to clearly identify positive and negative value of tool data.

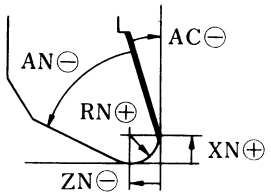


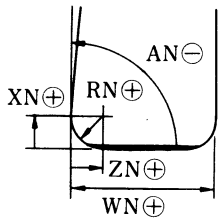
- Tool figure

  Input the dimensions, referring to the following tool figures, irrespective of whether the outer diameter cutting or inner diameter cutting is done. Assume that a tool was placed on paper with its setting face to an NC lathe turned upward, regardless of the front face or rear face of the cutting tool.

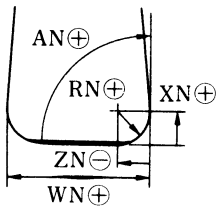
  The "AC" plus direction is counterclockwise around the major cutting edge as the center, while the "AN" plus direction is clockwise around the major cutting edge as the center. (The bold line in the figure shows the major cutting edge.)



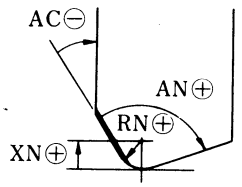




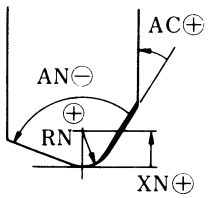
(Note)
AC = 90
AN = −90



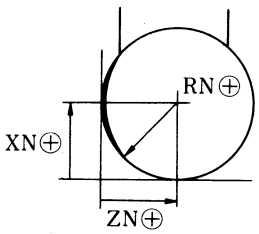
(Note)
AC = −90
AN = 90



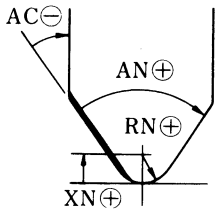
(Note)
ZN = 0



(Note)
ZN = 0



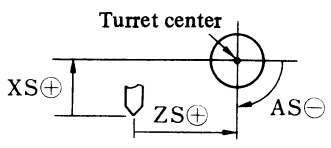
(Note)
AC = −0.02
AN = 0.04

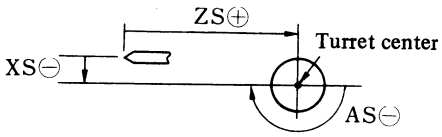


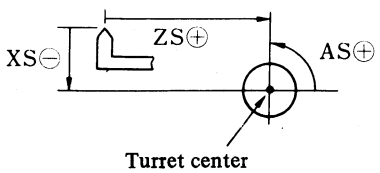
(Note)
ZN = 0

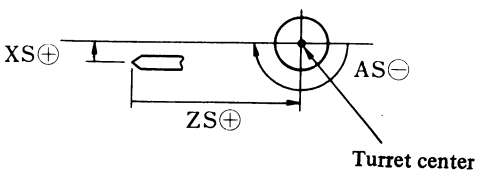
- Setting method

  "AS", "ZS" and "XS" indicate the setting direction and setting position of the cutting tool defined by the previous method. Determin the "AS" value and sign by turning the cutting tool in an optional direction as it is. "AS" angle is minus in the clockwise direction.



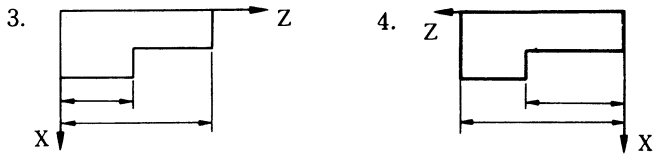






- When the tool post is set at the front side

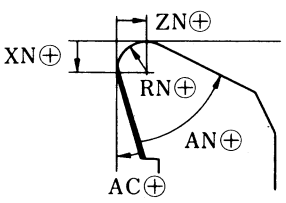
  In this case, select [3] or [4] as the drawing format in order to clearly identify positive and negative value of tool data.

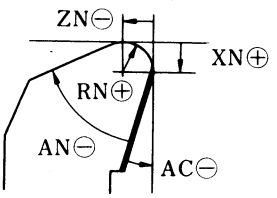


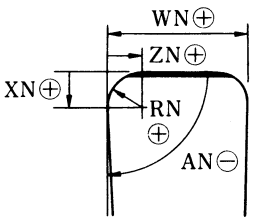
- Tool figure

  Input dimensions, referring to the following figures, irrespective of whether the outer diameter cutting or inner diameter cutting is done. Assume that a tool was placed on paper with its mounting face to an NC lathe turned upward, regardless of the front face or rear face of the cutting tool.

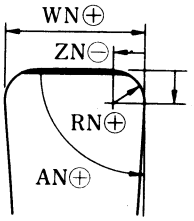
  The "AC" plus direction is counterlockwise around the major cutting edge as the center, while the "AN" plus direction is clockwise around the major cutting edge as the center. (The bold line in the figure shows the major cutting edge.)



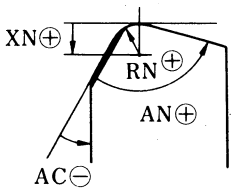




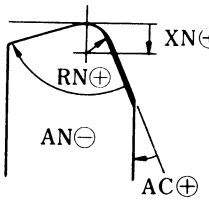
(Note)
AC = 90
AN = −90



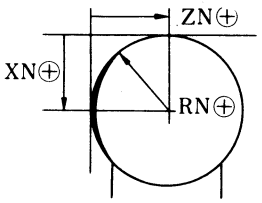
(Note)
AC = −90
AN = 90



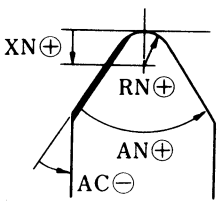
(Note)
ZN = 0



(Note)
ZN = 0



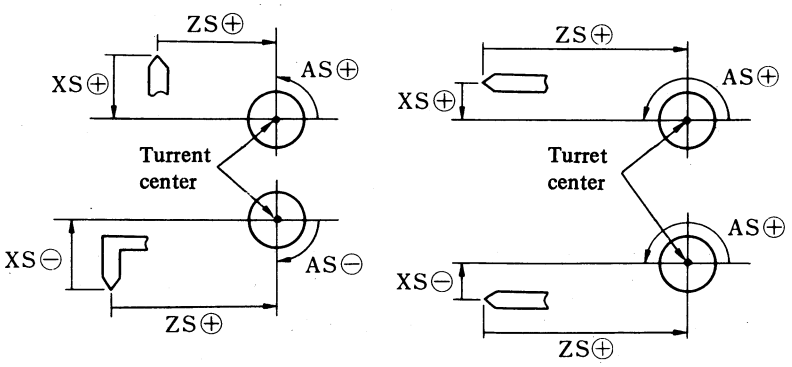
(Note)
AC = −0.02
AN = 0.04



(Note)
ZN = 0

- Setting method

  "AS", "ZS" and "XS" indicate the setting direction and setting position of the cutting tool defined by the previous method. Determin the "AS" value and sign by turning the cutting tool figure in an optional direction as it is. The "AS" angle is minus in the clockwise direction.



(Machining start position)

When all tool data are input, the system ask you the machining start position as follows.

MACHINING START POSITION

X-AXIS	**DX0** =	X-axis coordinate value
Z-AXIS	**Z0** =	Z-axis coordinate value

(Cutting conditions)

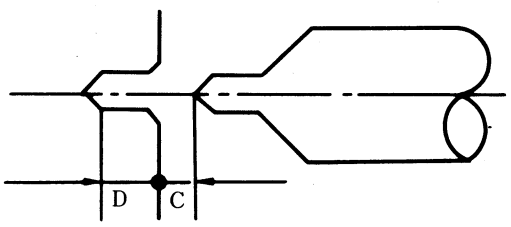
The system asks you the cutting conditions required for the machining data already selected in the CRT screen for determining the machining process. If a material file is already loaded, the cutting conditions are automatically set and displayed according to the kinds of selected materials. Change only the desired items, while monitoring the CRT screen. If you want to change the feedrate to 75% of the displayed value, for example, multiply the displayed value by 0.75 as follows.

(Example) F1 = [3] [*] [0] [.] [7] [5] [SP]

The cutting conditions and meanings of the system questions are described below according to machining data.

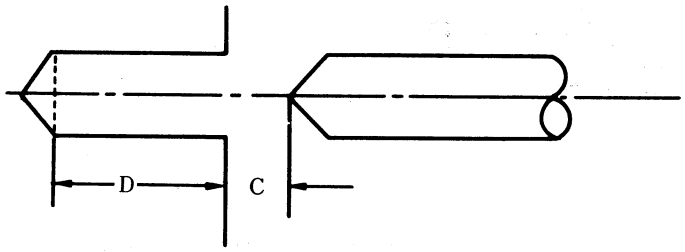
(1) Center drilling conditions

CLEARANCE	**C** =	Clearance quantity (mm)
DEPTH OF CUT	**D** =	Depth of cut (mm)
SPINDLE SPEED	**N** =	Spindle speed (rpm)
FEED RATE	**F** =	Feedrate (mm/rev)



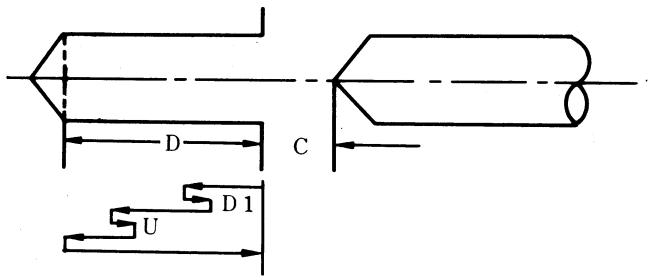
(2) Drilling conditions

CUTTING CONDITIONS:	**TYPE 1**	Cutting conditions type 1
CLEARANCE	**C** =	Clearance quantity (mm)
DEPTH OF CUT	**D** =	Depth of cut (mm)
SPINDLE SPEED	**N** =	Spindle speed (rpm)
FEED RATE	**F** =	Feedrate (mm/rev)

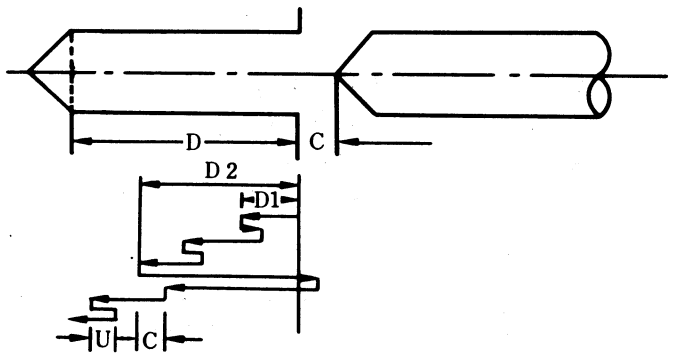


(Note 1) Cutting conditions for drilling comprise type 1, 2 and 3 according to the movement of a drill.

CUTTING CONDITIONS:	**TYPE 2**	Cutting conditions type 2
CLEARANCE	**C** =	Clearance quantity (mm)
DEPTH OF CUT	**D** =	Depth of cut (mm)
	D1 =	Depth of cut (mm)
RETURN AMOUNT	**U** =	Return amount quantity (mm)
SPINDLE SPEED	**N** =	Spindle speed (rpm)
FEED RATE	**F1** =	Feedrate (mm/rev)

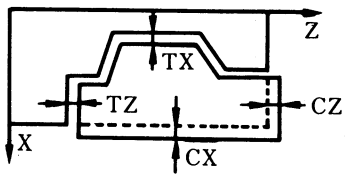


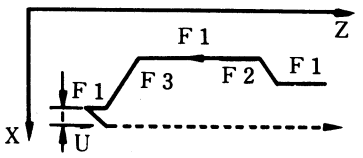
CUTTING CONDITIONS:	**TYPE 3**	Cutting conditions type 3
CLEARANCE	**C**	Clearance quantity (mm)
	C1	Clearance quantity in 2nd drilling (mm)
DEPTH OF CUT	**D**	Depth of cut (mm)
	D1	Depth in 1st drilling (mm)
	D2	Depth in 2nd drilling (mm)
RETURN AMOUNT.....	**U**	Return amount quantity (mm)
SPINDLE SPEED	**N**	Spindle speed (rpm)
FEED RATE	**F1**	Feedrate (mm/rev)
	F2	Feedrate in redrilling (mm/rev)



(3) Roughing conditions of outer and inner figure

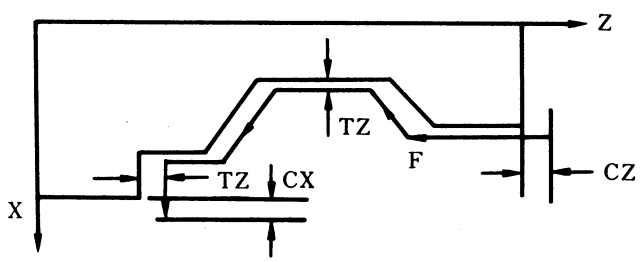
CLEARANCE	**CX** =	Clearance quantity in X-axis (mm)
	CZ =	Clearance quantity in Z-axis (mm)
FINISH ALLOWANCE	**TX** =	Finish allowance in X-axis (mm)
	TZ =	Finish allowance in Z-axis (mm)
DEPTH OF CUT	**D** =	Depth of cut (mm)
RETURN AMOUNT	**U** =	Return amount quantity (mm)
CUTTING SPEED	**V** =	Cutting speed (m/min)
FEED RATE	**F1** =	Feedrate (mm/rev)
	F2 =	Feedrate (mm/rev)
	F3 =	Feedrate (mm/rev)





(4) Semi-finishing conditions of outer and inner figure

CLEARANCE	**CX** =	Clearance quantity in X-axis (mm)
	CZ =	Clearance quantity in Z-axis (mm)
FINISH ALLOWANCE	**TX** =	Finish allowance in X-axis (mm)
	TZ =	Finish allowance in Z-axis (mm)
CUTTING SPEED	**V** =	Cutting speed (m/min)
FEED RATE	**F** =	Feedrate (mm/rev)

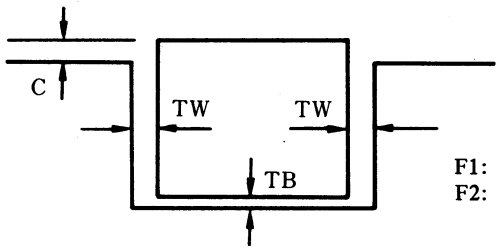


(5) Finishing conditions of outer and inner figure

CLEARANCE	**CX** =	Clearance quantity in X-axis (mm)
	CZ =	Clearance quantity in Z-axis (mm)
CUTTING SPEED	**V** =	Cutting speed (m/min)
FEED RATE	**F1** =	Feedrate for ▽ (mm/rev)
	F2 =	Feedrate for ▽▽ (mm/rev)
	F3 =	Feedrate for ▽▽▽ (mm/rev)
	F4 =	Feedrate for ▽▽▽▽ (mm/rev)

(6) Grooving and necking conditions

CLEARANCE	**C** =	Clearance quantity (mm)
FINISH ALLOWANCE	**TW** =	Finish allowance in width (mm)
	TB =	Finish allowance in depth (mm)
CUTTING SPEED	**V** =	Cutting speed (m/min)
FEED RATE	**F1** =	Feedrate in roughing (mm/rev)
	F2 =	Feedrate in finishing (mm/rev)

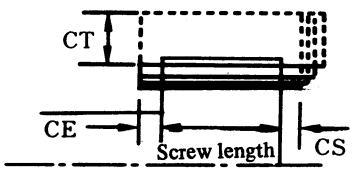


F1: Feedrate at roughing
F2: Feedrate at finishing

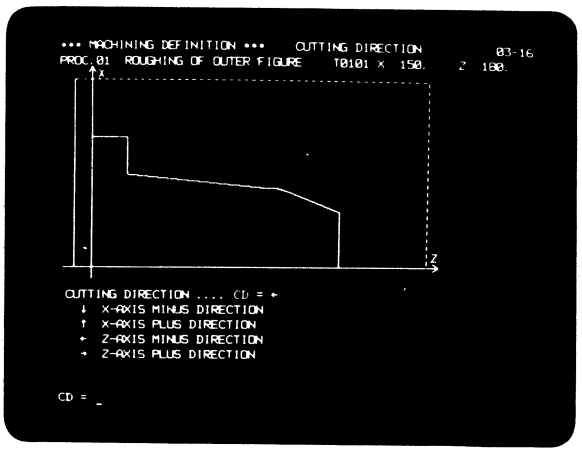
(Note) "TW", "TB" and "F2" are not asked by the system in necking condition.

(7) Threading conditions

CLEARANCE	**CT** =	Clearance quantity (mm)
	CS =	Clearance quantity (mm)
	CE =	Clearance quantity (mm)
DEPTH OF CUT	**D** =	Depth of cut (mm)
	DL =	Final depth of cut (mm)
CUTTING SPEED	**V** =	Cutting speed (m/min)



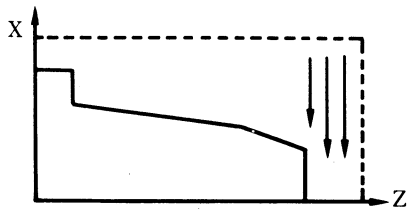
Cutting direction



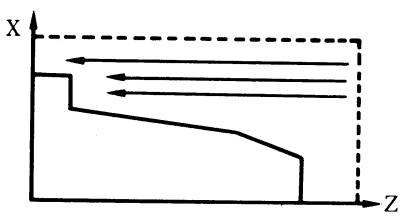
Menu

The system asks you a question to define the cutting direction. Input the cutting direction by arrow key.

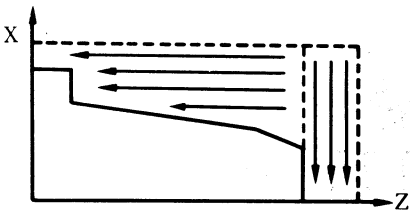
CUTTING DIRECTION	**CD =**	Cutting direction
↓	**X-AXIS MINUS DIRECTION**	X-axis minus direction
↑	**X-AXIS PLUS DIRECTION**	X-axis plus direction
←	**Z-AXIS MINUS DIRECTION**	Z-axis minus direction
→	**Z-AXIS PLUS DIRECTION**	Z-axis plus direction



Input [↓]



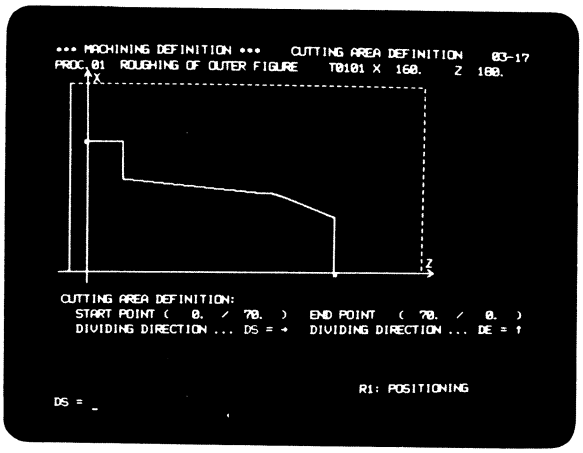
Input [←]



When the cutting directions differ in the end face and outer diameter, input [↓] if cutting is started with the end face.

Cutting area definition

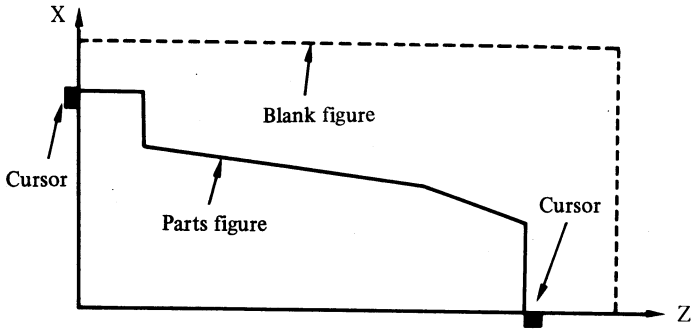
The cursors (■) are displayed along the parts figure to specify the cutting area.



A blank figure (dotted line) and machining figure (solid line) will be drawn on the CRT screen, the system asks you the dividing directions.

- Machining area

  Designate the cutting area of parts figure from the start point to end point by flickering cursors on the CRT screen. One cursor shows the start point, while the other shows the end point of the cutting area.



  Shift the cursor along the parts figure for selecting the start point and end point.

- Movement of cursor

  Depress [R1] **key.**

  The cursor shifts, each time [R1] key is depressed.

  It shifts forward or backward along the parts figure according to whether [F0] key is turned off or on at that time.

- Division

  After designating the start point and end point, input the dividing direction of the blank to define the cutting point.

  Depress arrow key when the system asks you the following questions.

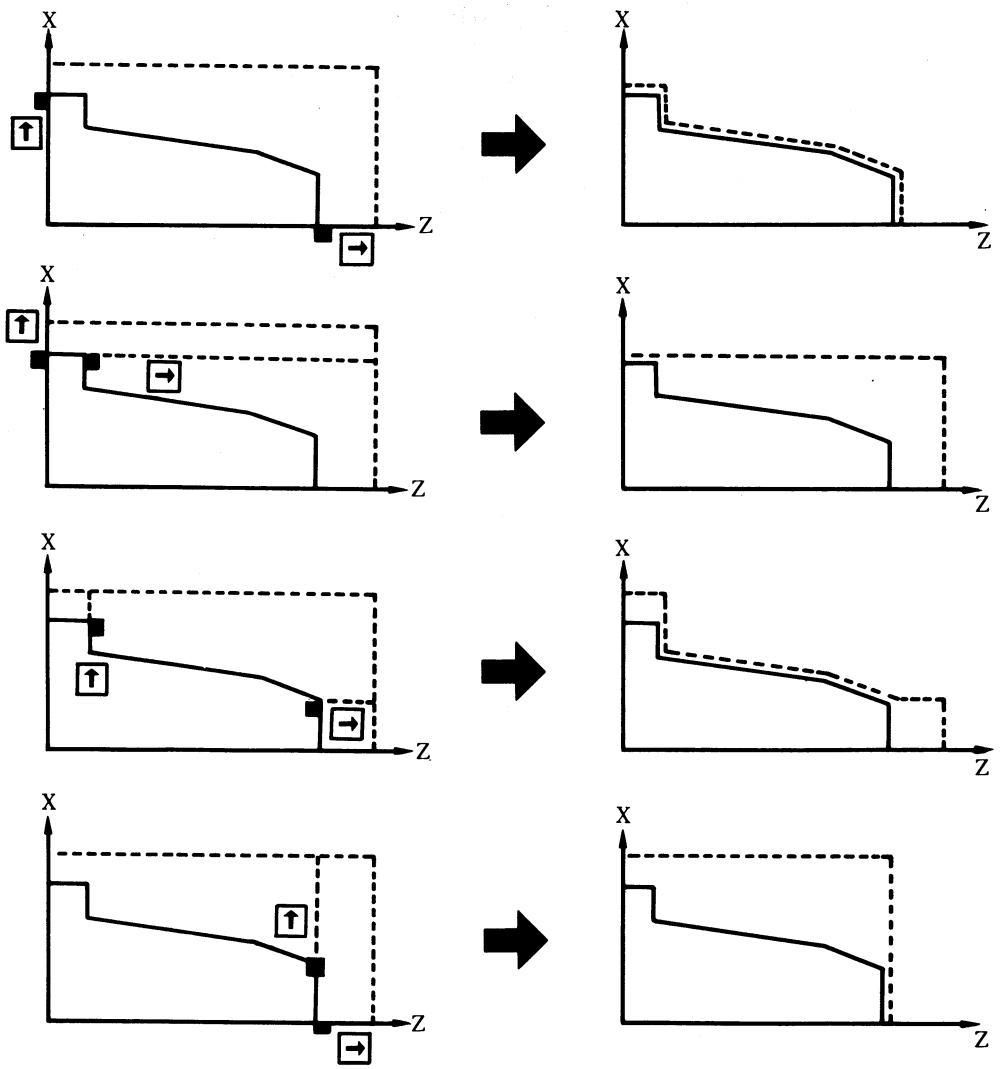
DIVIDING DIRECTION	**DS** =	Dividing direction at start point
DIVIDING DIRECTION	**DE** =	Dividing direction at end point

- Example of division

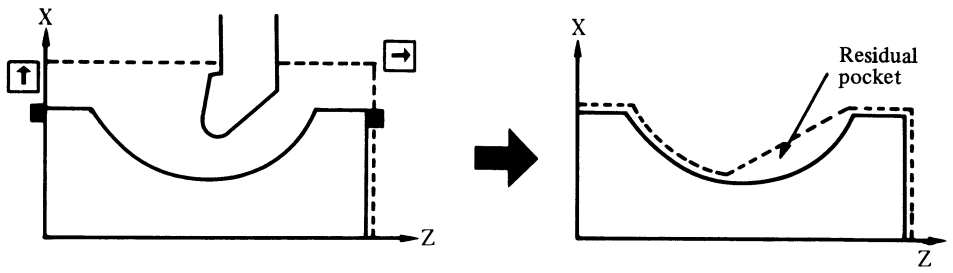
  Assume a line is drawn in the arrow direction from the flickering cursor when each arrow key is depressed. The portion bounded by the lines in these arrow directions, blank figure, and parts figure are machined.

  A residual blank figure, which may be kept unmachined after the cutting area has been lathed out, will be displayed on the CRT screen. Since Symbolic FAPT can display drawings of these residual blank figures one by one, you can check the remaining cutting quantity very easily.

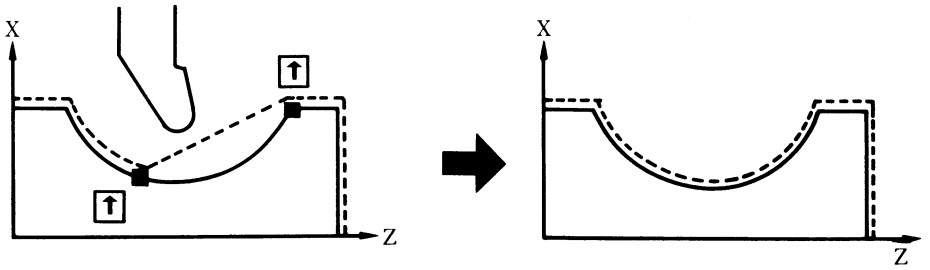
  It is possible that a part of blank cannot be machined by any means because of tool figures. In such a case, the system automatically checks the relation between tool figure and parts figure, and displays such an unmachined pocket on the CRT screen. Accordingly, you can machine the pocket by using another tool.



- Designation of cutting of residual pockets.



A pocket may be produced, depending upon parts figure and tool figure as illustrated above. In such a case, designate the area to cut such a pocket only by using a reversible tool.



It should be carefully noted that if a reversed tool is not properly positioned or if tool figure data is not appropriate, the pocket area cannot always be inputted.

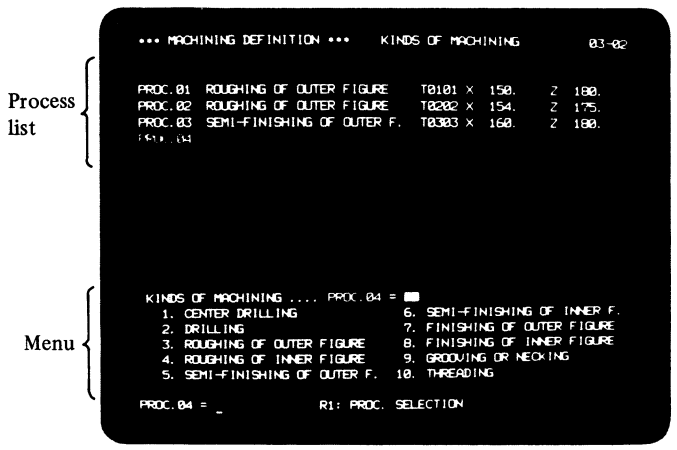
After the cutting area has been designated, the designated cutting area is lathed out from the blank figure by the tool. Then, the system asks you to check if another portion is to be cut by the same tool or not.

CUTTING ANOTHER AREA BY SAME TOOL ? ..... CN =
(1: YES 0: NO)

Answer this question by one of the following operations.

1	SP	..... In case of "YES". (Another position is cut by the same tool)
0	SP	..... In case of "NO". (Not cut by the same tool)

After answering all questions, the CRT screen display returns to the process list which asks you the kind of machining.

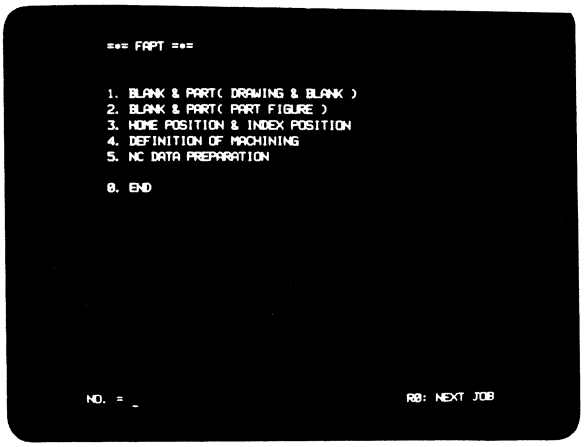


If you want to prepare NC data after defining the kind of machining, depress [SP] or [R0] key in responce to the system question about the kind of machining.

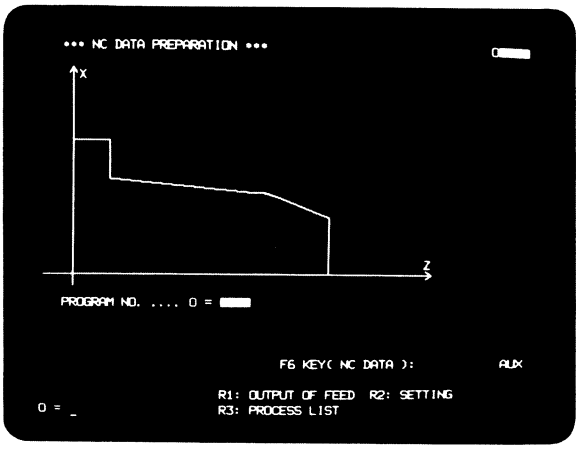
 [SP] or [R0]

3.5 NC Data Preparation

An execution sequence selection menu will appear on the CRT screen, and menu number [5] for NC data preparation will flicker.



 [SP] or [5] [SP]

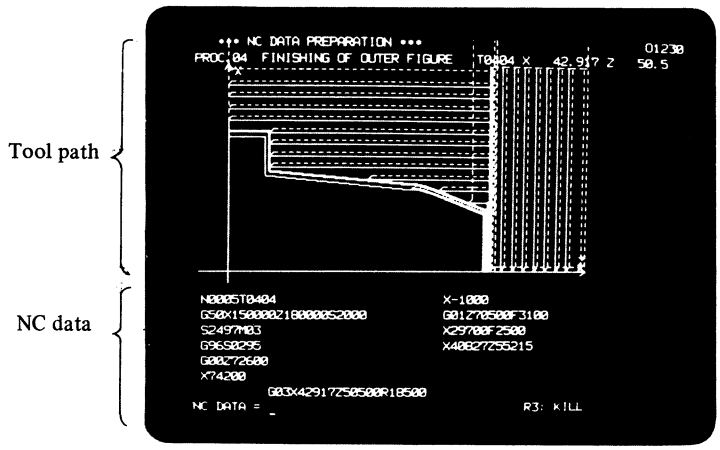


The system asks you program number as displayed above.

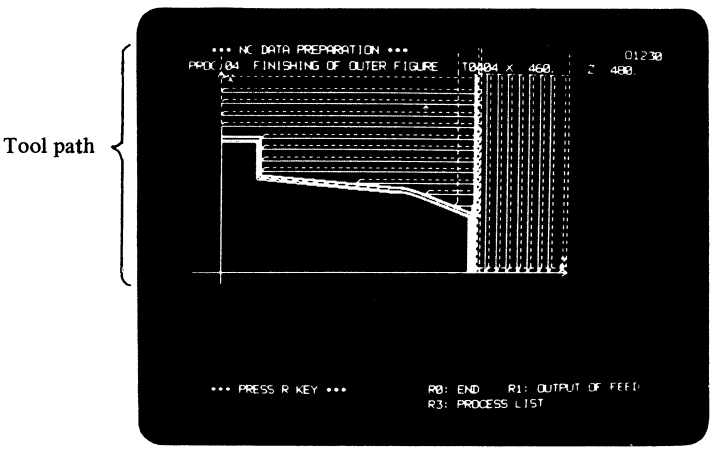
PROGRAM NO. ..... O =

PROGRAM NO. SP **and** R0

NC data are prepared in the sequence according to the process list, and a tool path is drawn on the CRT screen.



The "tool path" and "NC data" are displayed when F3 **key is turned on.**



The "tool path" only is displayed when F3 **key is turned off.**

(Note 1) For the method of using the F keys and selecting the output unit during NC data preparation, refer to the next chapter.

Display of machining time

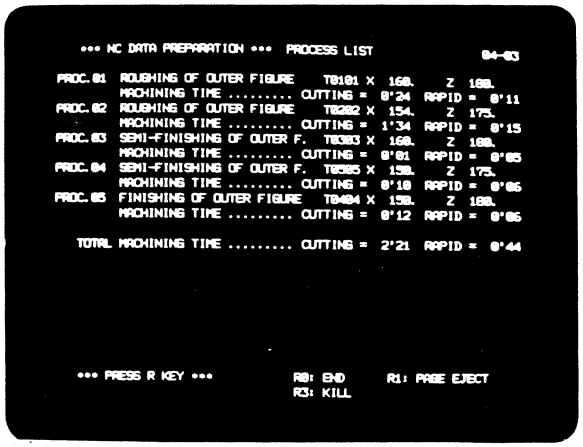
After NC data have been automatically prepared, a menu appears at the lower part of the CRT screen to select [R] keys to be depressed.

R0: END R1: OUTPUT OF FEED
R3: PROCESS LIST

When [R3] key is depressed, machining time for each tool employed and the total machining time are displayed as a list.

Depress [R3] key.

Machining time list



These machining time values indicate theoretical values required when NC data were automatically prepared.

If the feedrate override is applied during practical machining, the machining time varies resultantly.

F keys in NC data preparation

[F3] key

- When this key is turned on, automatically calculated NC data are displayed together with the tool path.
- When this key is turned off, the tool path only is displayed.

[F6] key

- When this key is turned on, NC data are output to the NC memory or memory cassette
- When this key is turned off, NC data are only displayed on the CRT screen.

[F7] key

- When this is turned on, NC data prepared every block, each time [SP] key is depressed. In this case data (M code, G code, NC data, etc.) are optionally inputtable from the keyboard. Also the NC data to be output next are displayed by one block at the lower part of the CRT screen, so that these data will be convenient for inserting optional data.
- When this key is turned off, NC data are output continuously.

3.6 Change of input unit (mm/inch)

This function applies to the system version E and higher.

In Symbolic FAPT, the input data unit is coincide with the output NC tape unit. If a blank dimension or a parts figure was input in the mm unit, for example, all other data should be input in the mm unit. Also, an NC tape is output in the mm unit.

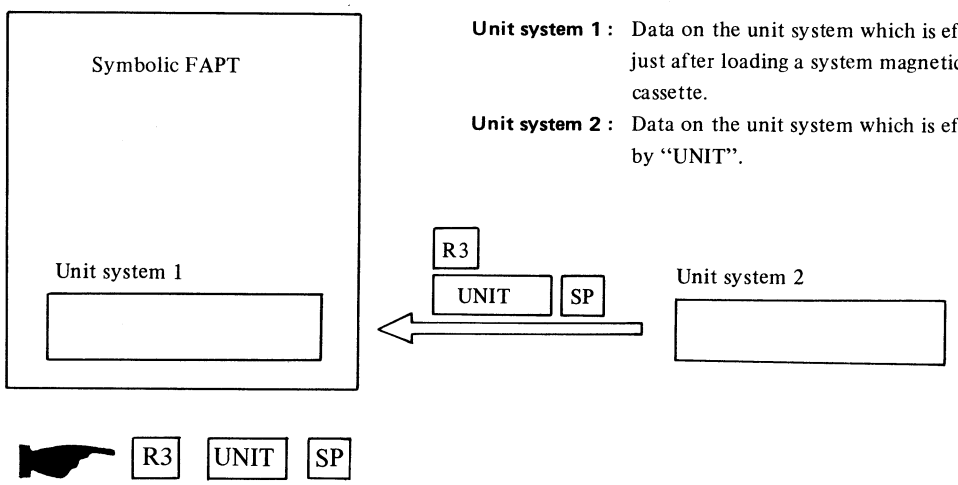
However, if a drawing is written in the inch unit and employed in a mm system machine tool, you can prepare an inch unit NC tape by the Symbolic FAPT function and operate the machine tool by the inch/metric conversion function of the NC machine tool.

The following two kinds of system magnetic tape cassette are possible

① Metric system magnetic tape cassette just after loading a system tape

② Inch system magnetic tape cassette just after loading a system tape

By depressing the [R3] key and keying in "UNIT" just after loading respective tapes, the system is changed to be a unit system different from the loaded unit system.
Load the system tape again, if you desire to program by the original unit system.



Unit system 1 : Data on the unit system which is effective just after loading a system magnetic tape cassette.

Unit system 2 : Data on the unit system which is effectuated by "UNIT".

Caution

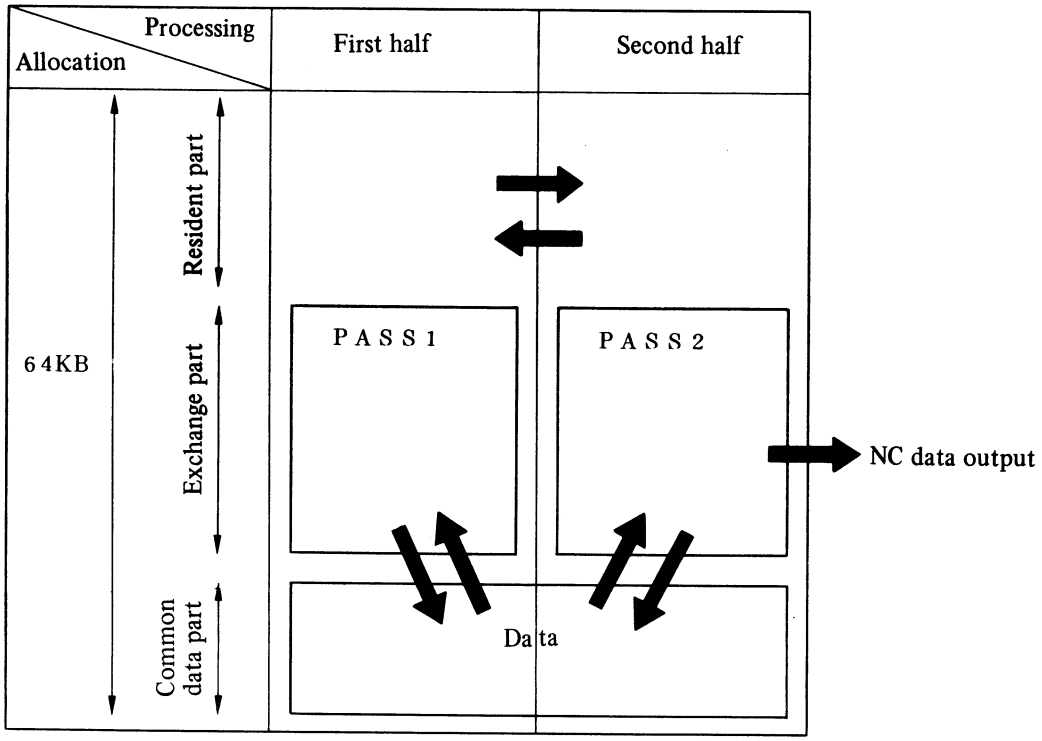
- The units of material data and tooling data are not automatically converted.
- Whenever of family program is input, the system unit system must be set to the unit system at the output time without fail before executing the program input.
- If the NC machine tool does not provide the inch/metric conversion function, enter dimensional values by utilizing the arthmetic function of Symbolic FAPT.
  For processing a drawing in the metric unit by using an inch system magnetic tape cassette (unit system of output NC tape is inch), for example, enter data as follows.
  d/25.4 [SP] (d: Dimension described in drawing)
- For the surface speed in the inch system, give data in the feed/min unit.

4. CAUTIONS WHEN USING Symbolic FAPT

4.1 Allocation of Memory

The memory capacity of the Symbolic FAPT processor (P-D) of the FANUC SYSTEM 3T-MODEL D is 64KB, while the Symbolic FAPT software capacity in the system program part and data part amounts to about 100KB in total. Since the Symbolic FAPT software is larger than the memory capacity, the Symbolic FAPT processing software cannot be executed by loading it into memory at once. Accordingly, it is executed by being divided into two parts. After executing its first half, the subsequent second half is executed by replacing the software.

The first half is called PASS1, while the second half is called PASS2.



Processing being contained in PASS1

- Symbolic FAPT execution procedure 1, 2, 3 (R0)
- Family program processing (R1)
- Input/output processing of various files (R2)
- MTF modification processing (R3)
- Material file

Processing being contained in PASS 2

- Symbolic FAPT execution procedure 4, 5 (R0)

4.2 Program Area Capacity

The capacity of the data part (user's program area) in the Symbolic FAPT is about 5.75KB, in which the blank figure file, part figure file, blank figure work file, pattern file, tooling file, and machining process file are sequentially produced.

When a process is defined, for example, an area required for the definition is to be searched and secured. If no unoccupied area is found, an error **"TEXT FILE OVER FLOW"** results.

If such a phenomenon occurred, the memory can be utilized effectively by the following methods.

(1) Don't use any incremental specification for figure definition (method of inputting a dimension by the increment from the present value when the system asks the dimension; method of affixing I to a numerical value)

When the definition of points was specified by the absolute value (by inputting the X and Z coordinate values), an 11-byte memory is required for loading one point.

On the other hand, if the definition of points was specified by the incremental value, 9 bytes are increased per coordinate value. In other words, when both X and Z coordinate values were specified by the incremental value, 29 bytes are required in total (18 bytes are increased).

Accordingly, if a message **"TEXT FILE OVER FLOW"** appears on the CRT screen, it is one of the countermeasures to define the figure again without incremental specification.

(2) Execute processing by dividing figures.

The definition areas for process, etc. increase, because the figure elements which occupy much memory decrease.

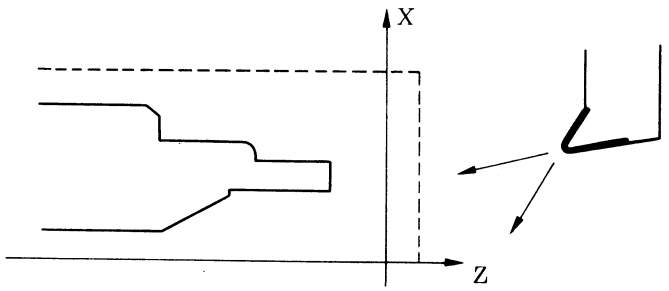
(3) Execute processing by dividing the process into "rough cutting" and "finish cutting" or higher stages.

Produce NC data for rough cutting first, and then, produce NC data by immediately defining the finish cutting in the new process definition. For the subsequent grooving, threading, and other machining, define the process separately like "finish plus grooving", "finish plus threading", and output desired NC data only without outputting finish process NC data.

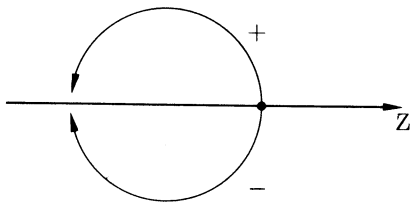
4.3 Tool Figure and Setting Method

If you know by what processing the tool figure and setting method are defined when tool data were input to the Symbolic FAPT system, you will be able to program these tools more securely.

Data employed by the system for checking the cutting direction, cutting edge angle, and other tool figures are angles indicating the tool nose directions in the NC data coordinate system.

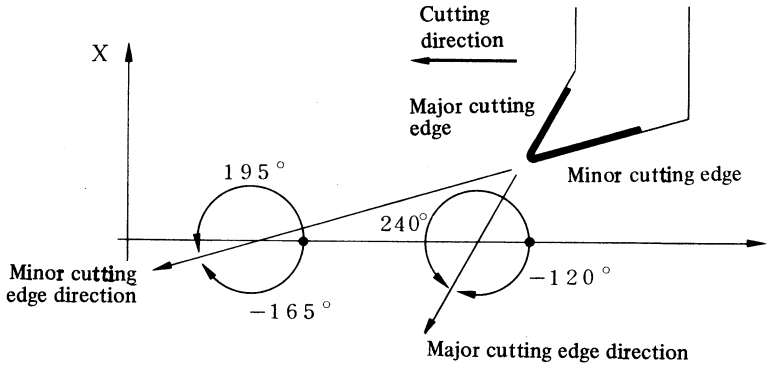


In the NC data coordinate system, the counterclockwise angle as viewed from the plus (+) Z direction shows a positive (+) angle, while the clockwise angle shows a negative (−) angle.



For example, 135° and −225° or 180° and −180° have the same meanings, respectively.

Two cutting edges are present normally. They are called major cutting edge and minor cutting edge. These cutting edge directions are called major cutting edge direction angle and minor cutting edge direction angle.

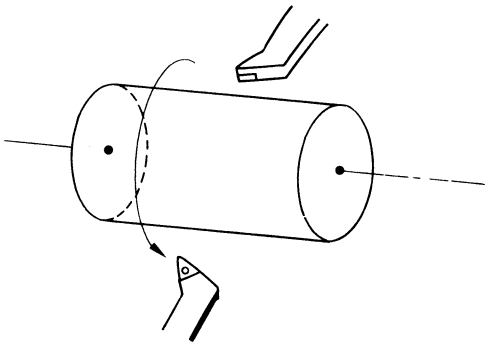


In the adove figure, the major cutting edge direction angle is 240° or −120°, while the minor cutting edge direction angle is −165° or 195°.

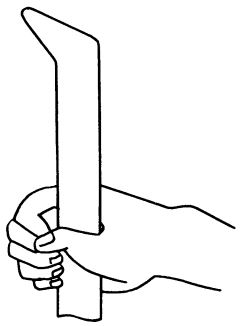
Now, the process of determining the tool figure and setting methods in the Symbolic FAPT will be described in regular sequence, referring to the above description.

1st step Positive direction and negative direction of input data

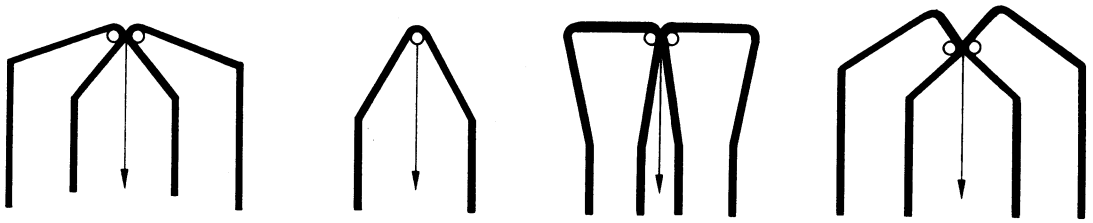
(1) The tool post is mounted on the front side or on the rear side, depending upon the types of NC lathes. If the tool post is mounted on the rear side, imagine a cutter located at a symmetric position on the front side, obtained by turning the cutter around the Z axis as the rotation center by a half turn.



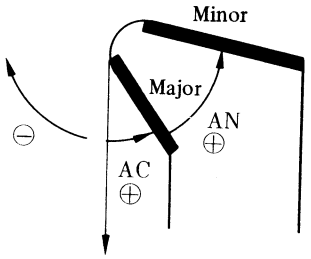
(2) Grip a tool while erecting its shank so that the cutting edge faces downward, if the cutter is mounted inside out, and the cutting edge faces upward in other cases.



Assume a straight line which passes the tool nose edge and faces just downward. The following figure shows examples typical figures of cutters and the straight line which passes the tool nose edge and faces just downward based on the presumption.



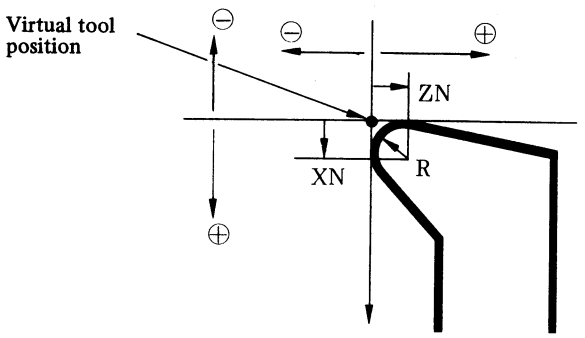
(3) The major cutting edge angle AC is defined as the angle formed by the major cutting edge angle and the straight line which passes the tool nose edge and faces just downward.
Cutting edge angle AN is also defined as the angle formed by the major cutting edge and minor cutting edge.
The counterclockwise angle is specified as the positive direction angle as shown in the following figure.



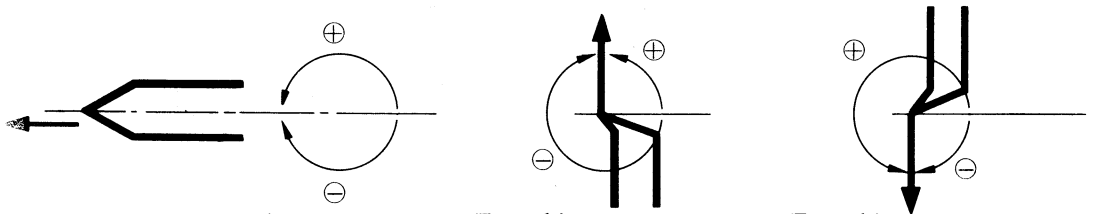
(4) If the straight line which passes the tool nose edge and faces just downward is presumed as a straight line which passes the virtual tool position, you will be able to understand the meanings of the following question asked on the CRT screen by the system.

VIRTUAL TOOL POS..... XN=
ZN=

Input the distance from the virtual tool position to the center of cutting edge as XN and ZN.



(5) Setting angle AS indicates the setting direction of a tool. Input the setting angle to the selected drawing format, irrespective of whether the tool post is located on the front side or on the rear side, and presume that the tool is set on the upper side of the CRT screen if menu number 1 or 2 was selected, or on the lower side of the CRT screen if menu number 3 or 4 was selected, for drawing format.
The upward direction of the straight line passing the tool nose edge employed in (2) (direction from the base of the tool to the tool nose) is employed for indicating the tool direction.
Input the clockwise direction as a negative (–) angle, from the right side of the abscissa of the CRT screen and also input the counterclockwise direction as a positive (+) angle.



2nd step Calculation of the direction angles of major cutting edge and minor cutting edge

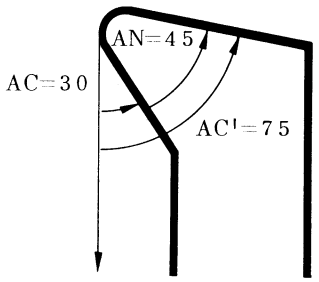
There are certain items to be input by an operator according to questions from the CRT screen.
The following three factors are used to determine the cutting edge direction angles.

CUTTING EDGE..... AC = (Cutting edge angle)
NOSE ANGLE..... AN = (Tool nose angle)
SETTING ANGLE.... AS = (Setting angle)

The following procedure shows the process of calculating the cutting edge direction angles inside the system based on the above three pieces of information.

(1) Assume AC be the major cutting edge angle and AN be the tool nose angle.

(Example) AC=30 AN=45



(2) Calculate the minor cutting edge angle by AC′ = AC + AN.

(Example) AC′ = 30 + 45 = 75

(3) Convert setting angle AS according to the selected drawing format. This is because that the system executes processing in the first quadrant, irrespective of the specification of drawing format.

- If the selected drawing format is 1 or 2, AS is kept unchanged.
- If the selected drawing format is 3 or 4, AS is converted into –AS (AS=–AS).

(4) Calculate the major cutting edge and minor cutting edge direction angles, respectively.

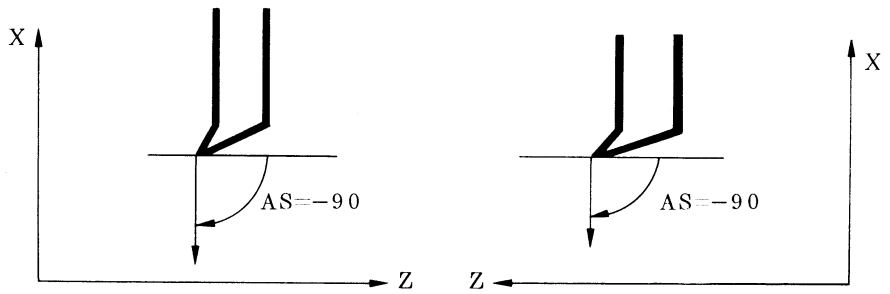
A = AS – AC (major cutting edge)

A′ = AS – AC′ (minor cutting edge)

Let's confirm, referring to the following examples, that the major cutting edge direction angle and minor cutting edge direction angle are obtained as desired, irrespective of the drawing format.

(Example 1) When drawing format is 1 or 2;

Input data: AC = 30, AN = 45, AS = –90

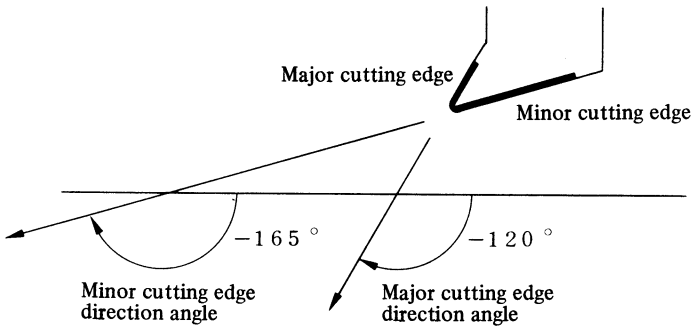


Drawing format 1 Drawing format 2

Internal processing; A = AS – AC = –120 . . . . . Major cutting edge direction angle

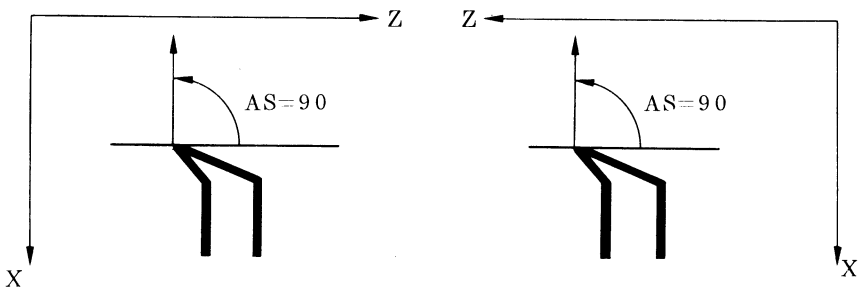
A′ = AS – AC′ = AS – (AC + AN) = –165 . . . Minor cutting edge direction angle

Results;



(Example 2) When drawing format is 3 or 4;

Input data; AC = 30, AN = 45, AS = 90

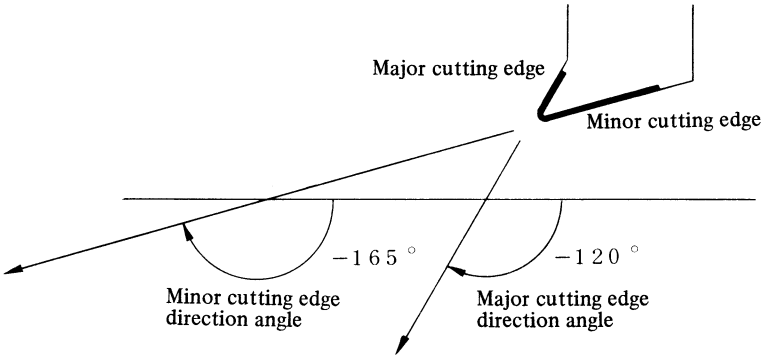


Internal processing; AS = −AS = 90

A = AS − AC = −120 . . . . . Major cutting edge direction angle

A′ = AS − AC′ = AS − (AC + AN) = −165 . . . Minor cutting edge direction angle

Results;

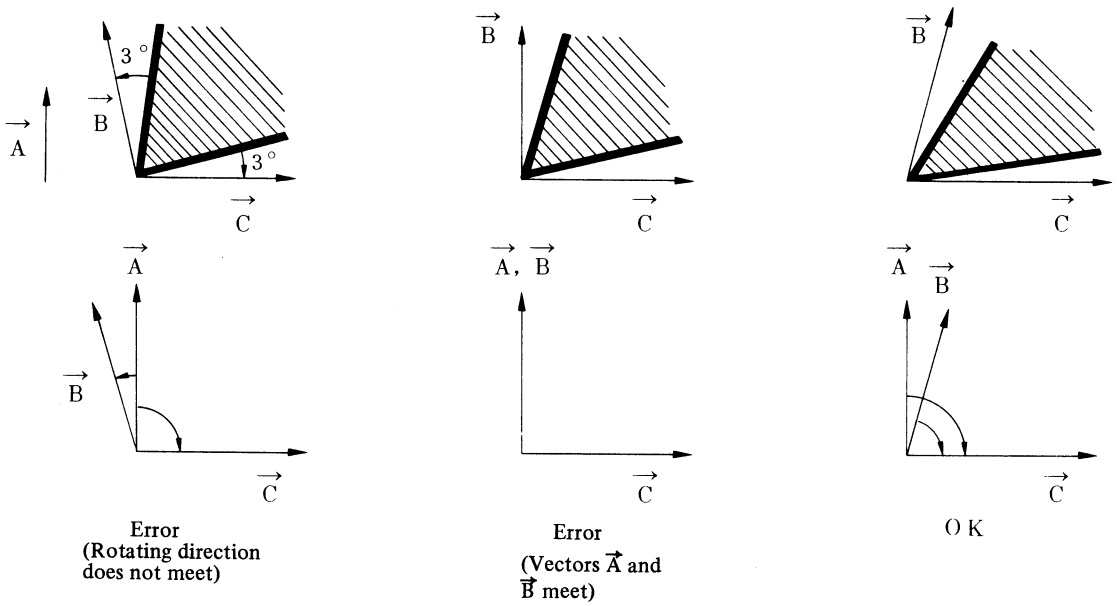


4.4 Cutting Direction and Cutting Edge Angle

For the cutting direction vector ($\vec{A}$) as viewed from the tool tip, and the cutting edge angle vector of major cutting edge ($\vec{B}$) and the cutting edge angle vector of minor cutting edge ($\vec{C}$) as viewed from the tool tip by taking the cutting edge protective angle (standard value: 3°) into consideration,

- an error is produced, if the rotating direction from $\vec{A}$ to $\vec{B}$ does not meet the rotating direction from $\vec{B}$ to $\vec{C}$, and also
- an error is produced if $\vec{A}$ meets $\vec{B}$.

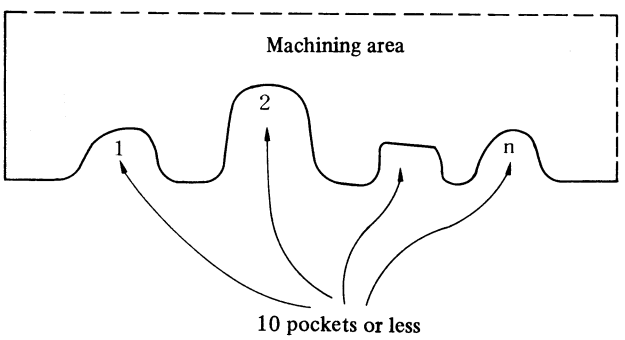
Example when the cutting direction is [↑];



When cutting is made at the minor cutting edge, consider the above explanation by exchanging vectors $\vec{B}$ and $\vec{C}$.

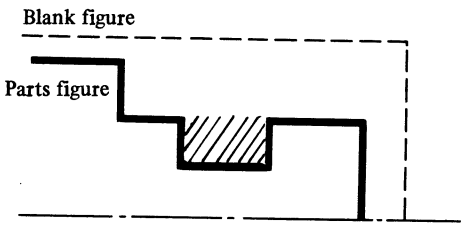
4.5 Rough Cutting Direction

Rough cutting is made by linear cutting (parallel to X axis or parallel to Z axis). Specify the medium finish cutting several times repeatedly, if an output of a profiling mold is desired. For number of pockets in rough cutting area, ten or less pockets are present in one machining area.



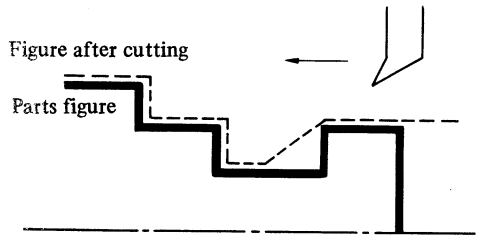
4.6 Judgement of Pockets

The pockets to be judged by the Symbolic FAPT are defined as "concaved parts as viewed from the cutting direction out of them". The same profile may be judged as a pocket or not judged as a pocket, depending upon the cutting directions as shown in the following example.

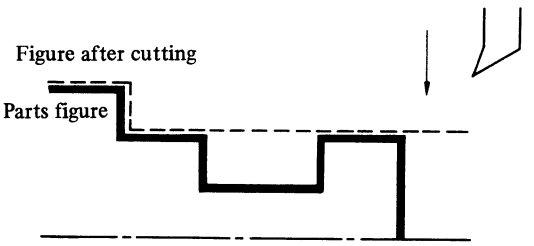


The concaved portion indicated by the slanting lines in the left figure is judged as a pocket when the cutting direction is [←] and it is not judged as a pocket when the cutting direction is [↓].
The concaved portion which was not judged as a pocket is not cut in the corresponding process.

When the above portion is cut in the [←] direction;



When the above portion is cut in the [↓] direction;



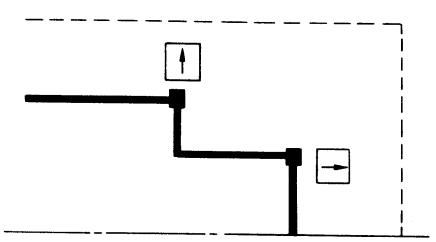
4.7 Division of Machining Area

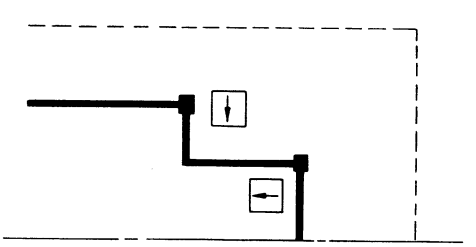
The arrows indicating the division direction are input for specifying the area based on the following principle.

Outer diameter side [↑]
Inner diameter side [↓]
Edge face side [↑] [↓] [→] (selective use)

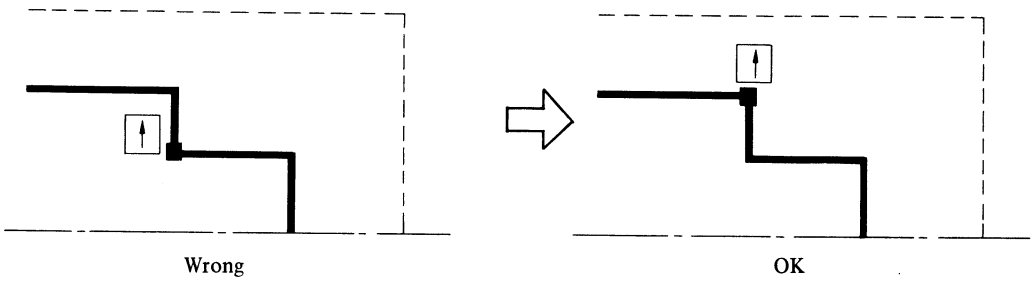
Refer to the following cautions when specifying the dividing directions of the machining area.

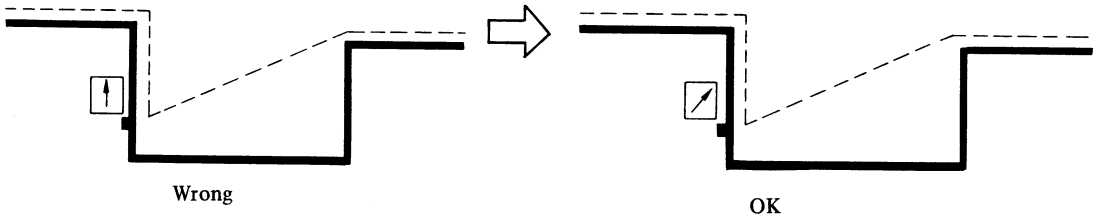
(1) Specify the dividing direction of the machining area toward the blank profile.



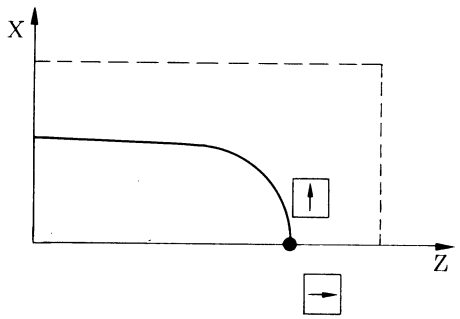


(2) Don't overlap the dividing direction of the machining area and the part figure each other.



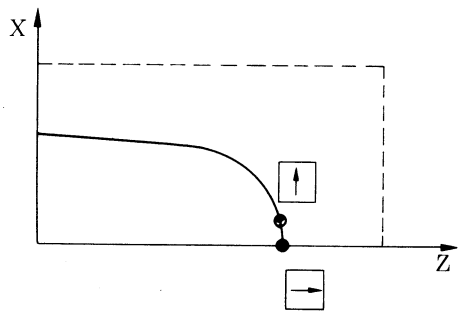


(3) No machining is possible, if the start point of part figure coincides with the end point and these points are located on a blank profile (blank figure).

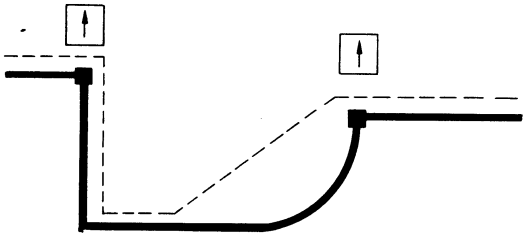


Z axis judges it as the profile figure of a blank.

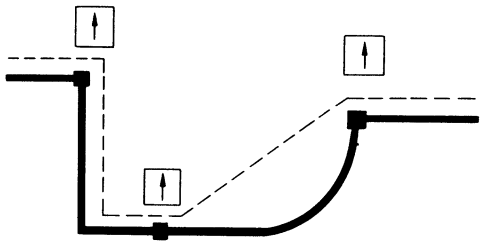
In case of the above figure, machining becomes possible, if a dummy figure element of about 0.01 mm in length is inserted.



(4) If a blank figure contains a concaved part, the blank must be machined by dividing the concaved part into two or more parts.



Wrong

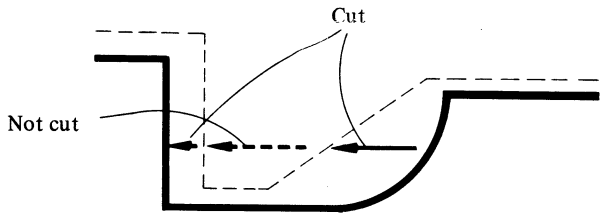


OK

The left figure shows an example when a blank in the machining area contains a concaved part. If this blank is attempted to be machined, non-cut portion being caught by two cutting parts (solid line parts in the figure shown below) will be produced in one tool pass.

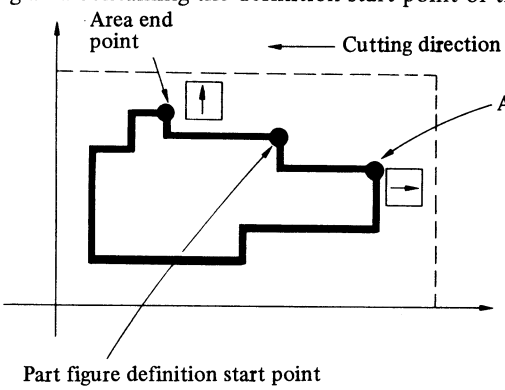
The Symbolic FAPT does not execute such processing as a blank on both sides of a tool pass is machined by crossing a space or the blank.

This problem can be solved by dividing the concaved part into two or more parts as illustrated in the above right figure.

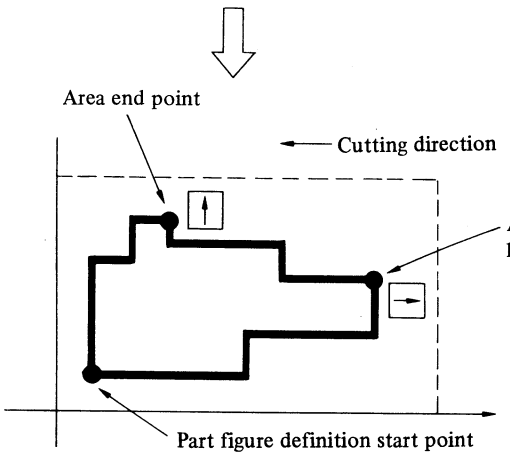


(5) Closed part figure

The cutting area containing the definition start point of the part figure cannot be specified.



Such an machining area as shown here cannot be specified.

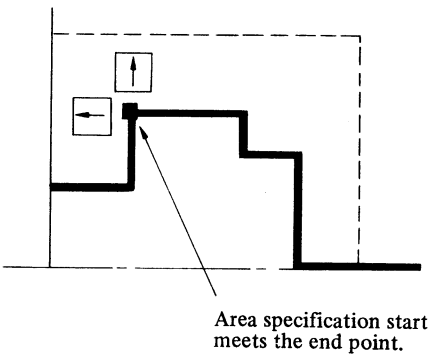


Select the left end part as the part figure definition start point.

It is allowable that either machining area start point or end point serves as the part figure start point.

(6) Cutting on the chuck side

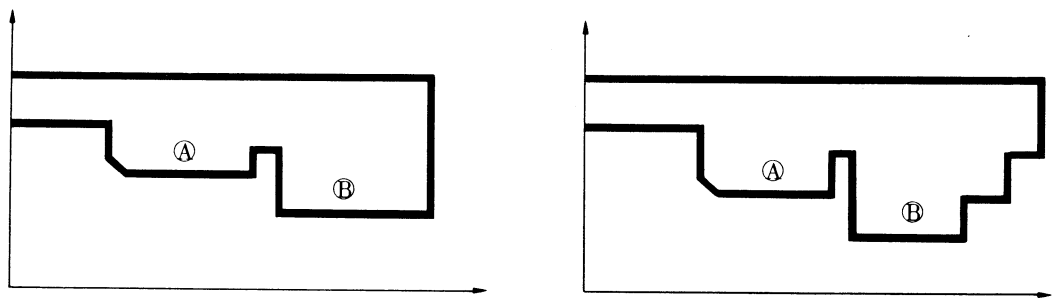
If the area specification start point meets its end point in such an area as the upper left end of a blank is contained on the machining area, this area cannot be specified, otherwise an error is produced or the blank figure is broken.



4.8 Approach and Return Relief to Inner Diameter

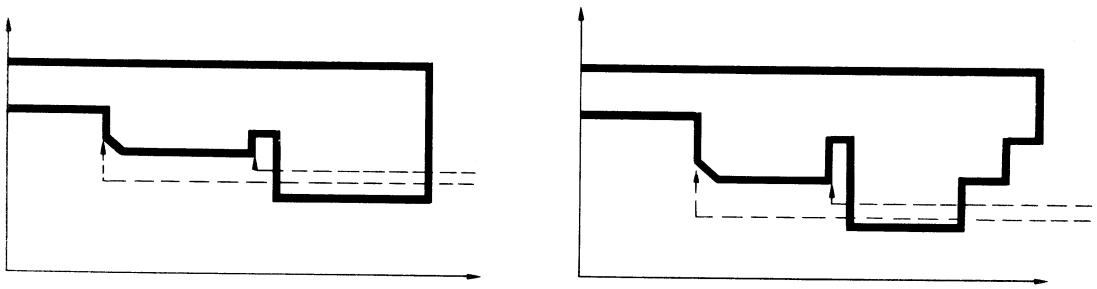
If an inner diameter contains a convexed part halfway, it cannot be machined because of the interference with the convexed part during approach or return relief.

Example of a figure containing a convexed part halfway.

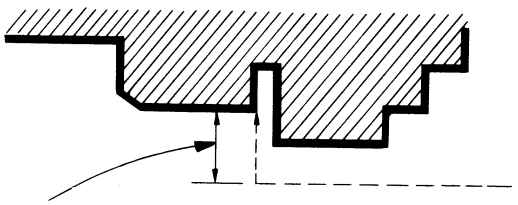


In the above figure, Ⓑ is an intermediate convexed part.

If grooving, threading, finish cutting, or rough cutting of part Ⓐ is attempted, it causes the interference with the convexed part.



Provided that the interference can be prevented during grooving by giving a clearance higher than the convexed part.



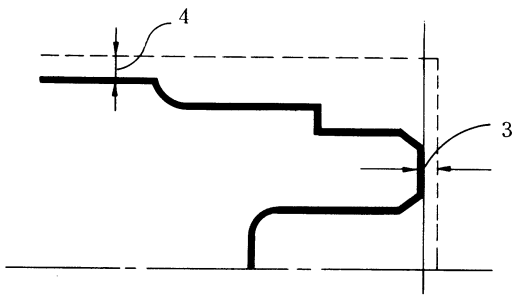
Clearance (To be higher than the convexed part)

4.9 Finish Allowance and Surplus Thickness

(1) Relation between finish allowance and surplus thickness

Don't give any finish allowance (in rough cutting and medium finish cutting) larger than or equivalent to the surplus thickness, otherwise the blank figure may be deformed to unfavorably affect the subsequent machining.

(Example)

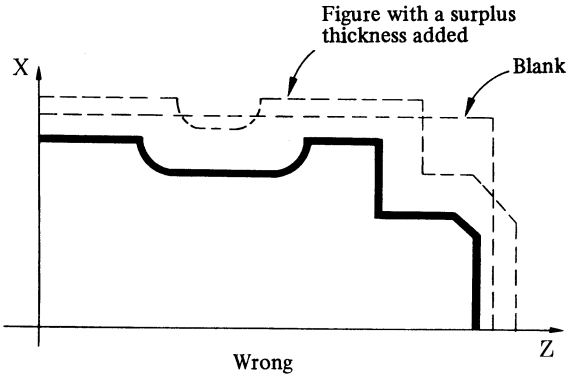


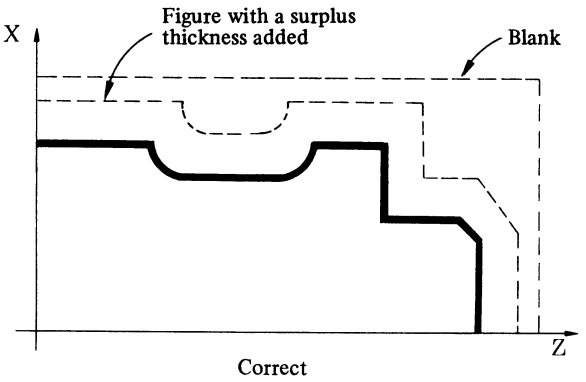
If the surplus thickness is as illustrated in the left figure, for example, the following finish allowances cannot be given.

Finish allowance in X direction 5mm
Finish allowance in Z direction 5mm

(2) Relation between surplus thickness and blank

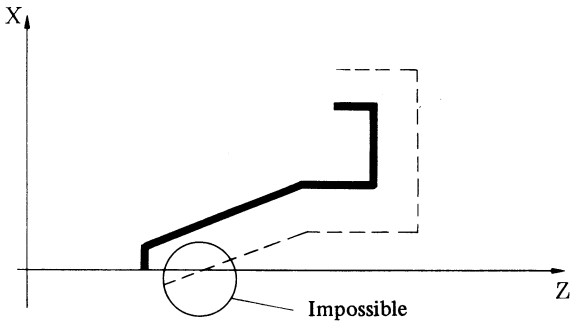
Don't give such a surplus thickness as may exceed a blank dimension in a special blank. It is recommended to give a relatively large blank dimension so that the surplus thickness does not exceed the blank figure, even if it is added to the machining figure.





(3) Relation between surplus thickness and Z axis

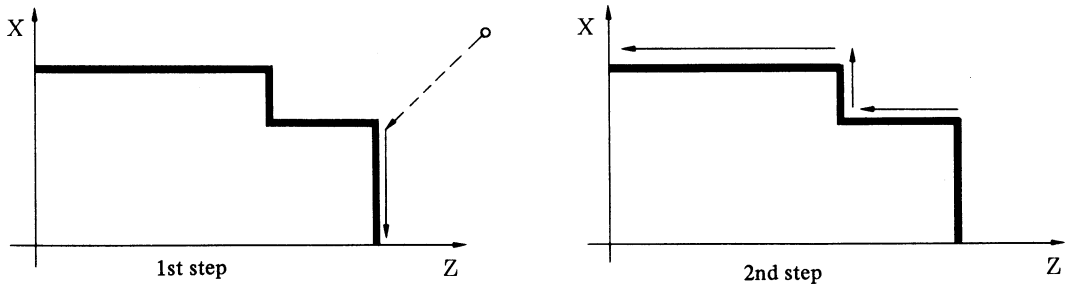
Since the Z axis is judged as a part of the blank figure in the system software, such a figure as an offset result using the surplus thickness exceeds the Z axis, cannot be machined.



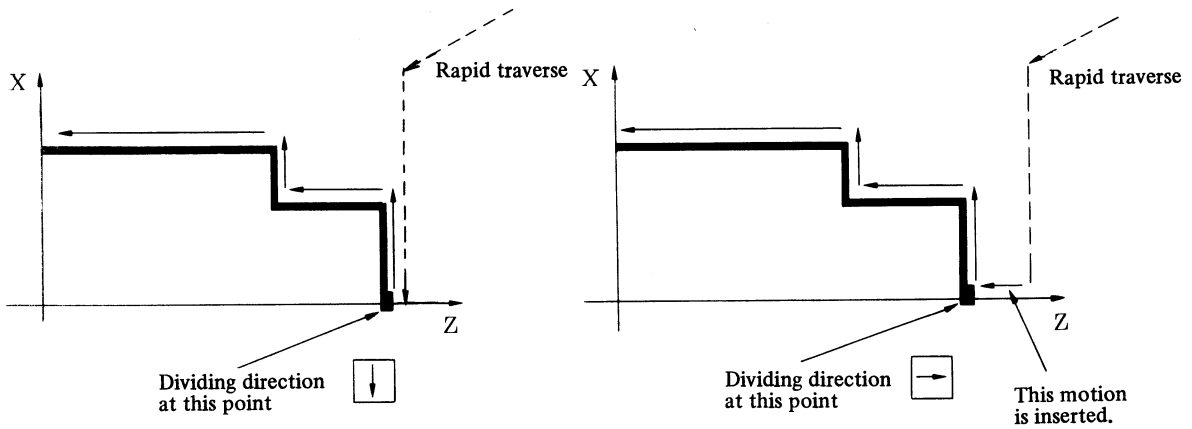
4.10 Semi-finish and Finish

(1) Dividing direction and clearance

If the end face finish and outer diameter finish are made by using the same tool, finish the end face before finishing the outer diameter, in principle.



Be careful not to specify a job to finish the end face and outer diameter continuously, otherwise the tool moves as follows.

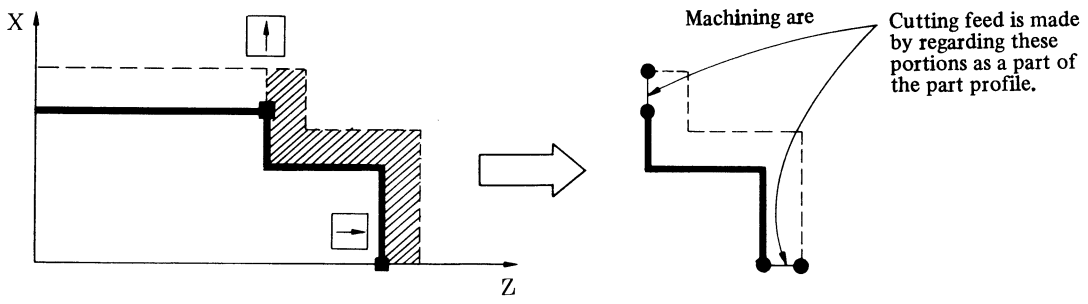


The specified clearance quantity may be inserted or not according to the dividing directions of the machining area.

Be careful since the tool moves toward a blank by the rapid traverse without any clearance quantity, if the dividing direction is as shown in the left figure.

(2) Concept of area in semi-finish and finish

In the semi-finish and finish, cutting is made along the part figure only, while neglecting the blank figure. The profile in the arrow direction specified when dividing the machining area is regarded as a part of the part profile.

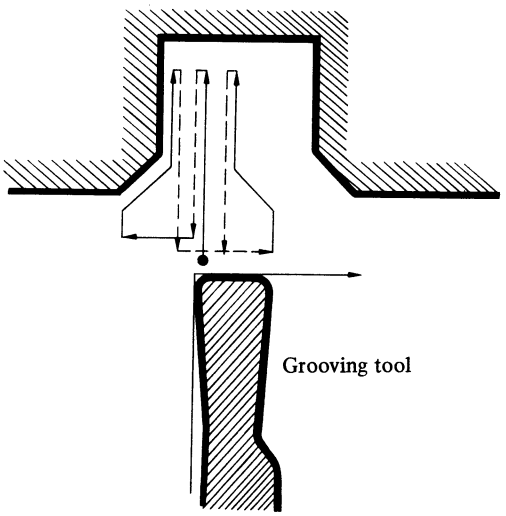


4.11 Grooving

(1) Tool path in grooving

Grooving is made by the following three steps.

- Apply the tool to the center of the groove.
- Drive the tool in such a manner as the groove is cut by the reference side cutting edge of the tool.
- Drive the tool in the opposite direction.



If the groove width is equal to the tool width, the tool is applied to the groove center once.

(2) Interference check

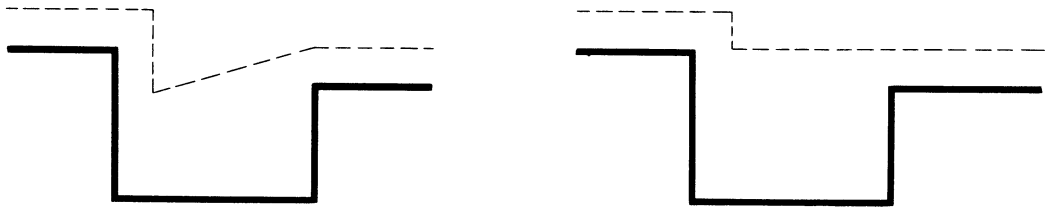
The interference check is not executed to check if the tool is normally applied to the groove or not.

Particularly be careful with the following points.

- The tool width should not be larger than the groove width.
- The groove should not be deeper than the tool length.

(3) Blank figure in grooving area

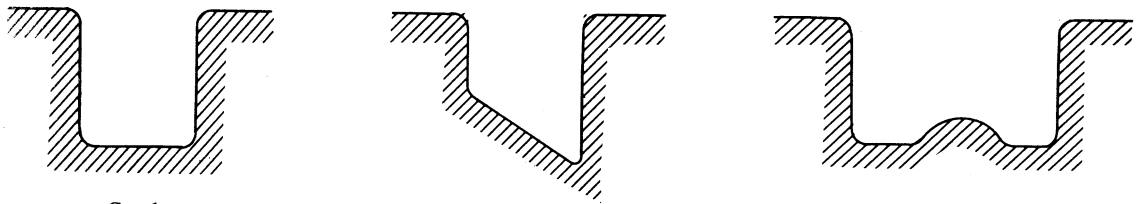
The blank figure at the entrance of the groove must be parallel to the X axis or Z axis in case of grooving.



Wrong OK

(4) Groove bottom figure

The groove bottom must be flat.

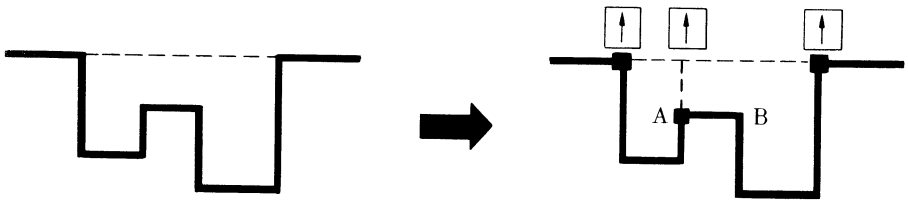


Good Wrong Wrong

The flat portion must be equal to or larger than the flat portion of the tool nose.

(5) Groove containing two or more concaved parts

If a groove contains two or more concaved portions, it cannot be machined by one process only. However, it can be machined by dividing it into two or more portions.

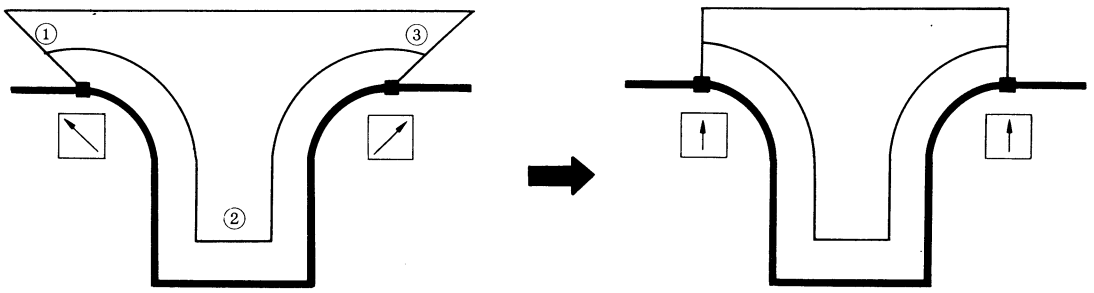


Cannot be grooved.

Grooving is possible when the groove is divided into A and B portions.

(Special example)

As a special case, grooving becomes impossible when two or more concaved parts exist in a groove, depending upon the dividing directions of the groove, even if the groove figure is normal.

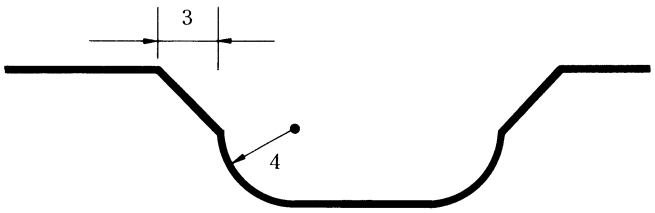


If ↖ and ↗ are specified as the dividing directions as illustrated in the above figure, three concaved parts ①, ②, ③ are produced in the grooving area, causing the grooving to be impossible.

If ↑ is specified as the dividing direction, only one concaved part exists, and grooving is possible without any trouble.

(6) Groove having no wall

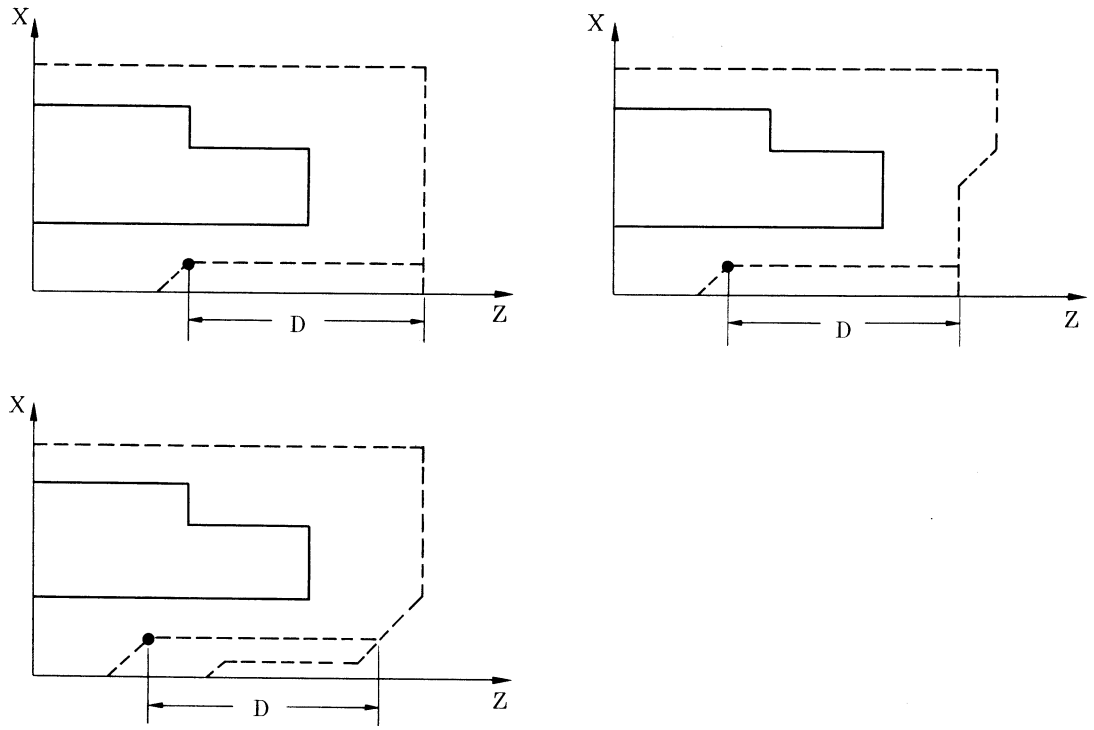
Grooving is impossible, if a groove wall is eliminated as a result of specifying C (chamfering) or R (corner R) at the groove entrance or bottom.



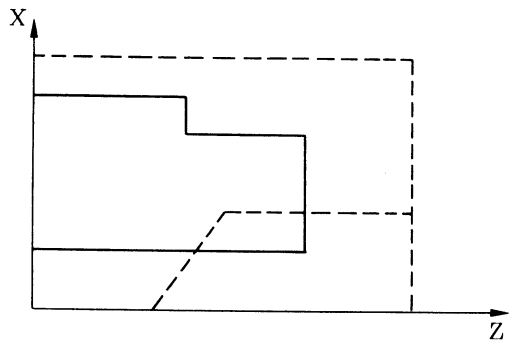
Groove depth : 7mm
C at entrance : 3mm
R at buttom : 4mm

4.12 Depth of Cut (D) in Drilling

The depth of cut is determined in the way that the drill shoulder is positioned at the place where the drill is driven by depth D from the intersection of the blank figure profile at that time (Blank figure changes as machining advances) and the tool diameter employed.



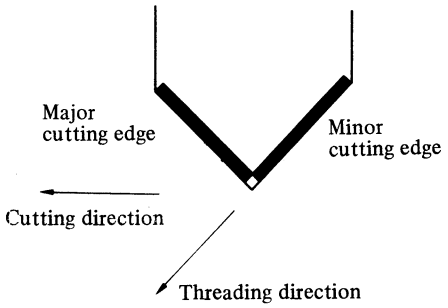
No intrusion to part is checked. Judge it, while monitoring the CRT screen.

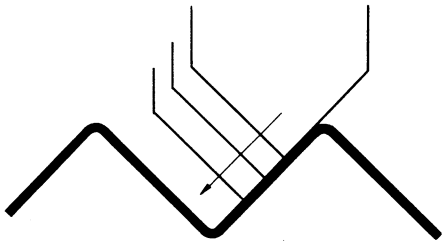


4.13 Threading

(1) Threading direction

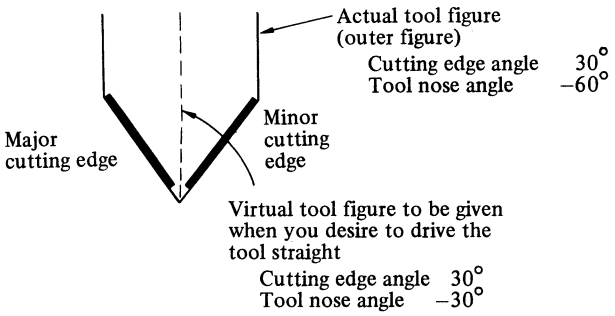
A tool is driven along an angle to which the minor cutting edge faces in threading.

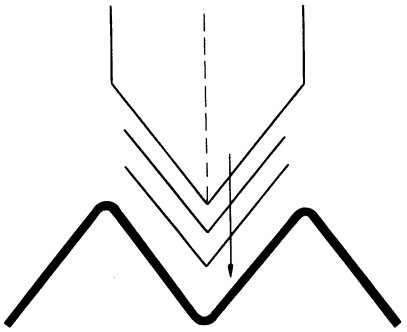




If you desire to determine the threading direction, irrespective of the tool figures, you have only to define it to allow the minor cutting edge to face the threading direction when defining the tool.

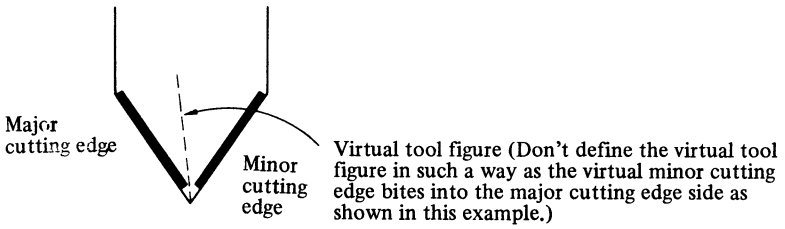
If you want to drive the tool straight, for example, you have to define the minor cutting edge angle so that the tool drives straight, irrespective of actual tool figure.





When defining the virtual tool figure, don't define it in such a way as the minor cutting edge bites into the major cutting edge side.

The allowable threading is limited up to the direction parallel to the X axis.



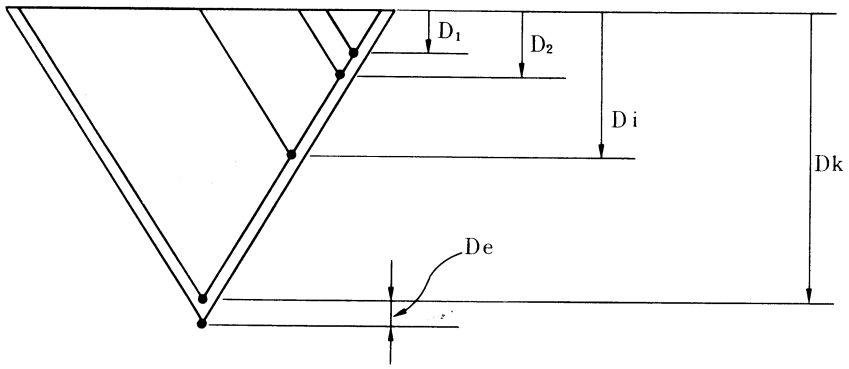
(2) Threading depth

D_1 : First threading depth (Absolute quantity: Material file)

D_i : i-th threading depth (Absolute quantity)

D_k : Threading depth once before the last

D_e : Last threading depth (Incremental quantity: system parameter No. 0119)



The threading depth is determined by the following equation.

$$D_i = D_1 * \sqrt{i}$$

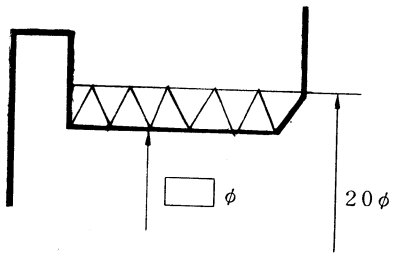
$$D_{i+1} - D_i \leqq D_e$$

Since D_k is not calculated by the above equation, it is possible that D_k becomes an increment smaller than De.

(3) Depth of screw thread

The depth of screw is calculated by pitch * constant 0.6495 (system parameter No. 0125: variable), and the minor diameter of screw is determined as a result. The external thread and internal thread are also calculated by the same method.

Accordingly, if the major diameter of internal thread is M20, for example, calculate the depth of screw from the pitch of screw and constant as the diameter for inputting the figure element.



(Example)

M20, P:Pitch

Minor diameter of internal thread is calculated by the following equation.

Inner dia. =20– (P X 0.64595)

4.14 S Code Output

(1) S code (2 digits)

S code (2 digits) can be output automatically.

For outputting it, observe the following procedure.

(a) Set MTF parameter number 1085 so that the constant surface speed control is not used.

(b) Change the output S code (4 digits) to an S code (2 digits) with the step feed during NC data output.

(2) S code value and output timing

The Symbolic FAPT is provided on the presumption that the NC machine tool is provided with the constant surface speed control function.

It entrusts NC (machine control unit) functions with the control to keep the surface speed constant during rough cutting, etc.

If the constant surface speed control is not provided or if this function is not used, it is necessary to change the MTF in advance so as not to use the constant surface speed control.

If such machining as changes the surface speed is made, an S code corresponding to the spindle speed calculated from the average revolution radius is output once.

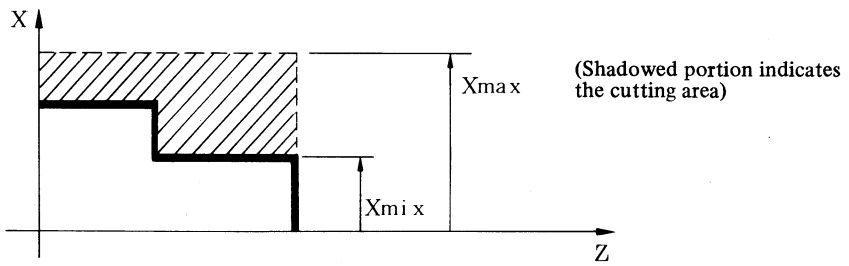
If cutting was made at plural points by using the same tool within the same process, the S code is not output to meet the cutting range in the second and higher points.

If a change of an S code is required, the S code should be inserted into corresponding place by setting the step feed at the NC data output time.

If plural points are cut by using the same tool within the same process when the constant surface speed control is employed, the S code is output based on the cutting speed specified by the first machining condition specification, and it is no longer output in the second and subsequent machining condition specification, even if the cutting speed was changed.

(3) Calculation of S code

The value of the S code to be output is determined as follows.



Assume that the cutting area is as illustrated above.

(a) Cutting with the constant surface speed control

- The spindle speed which is calculated by the specified surface speed and Xmax is output.
- The constant surface speed control mode is set after approach.

(b) Cutting without using constant surface speed control

- The spindle speed S which is calculated by the specified surface speed according to the following equation.

$$S = \frac{K \cdot V}{2\pi} \left(\frac{1}{Xmax} + \frac{1}{Xmin} \right)$$

In case of metric system: K = 1000

In case of inch system: K = 12

V : Surface speed

X max, X min: Mentioned in the above figure

- Cutting is made with the output S code hereafter.

If the S code calculated as specified above exceeds the maximum spindle speed (system parameter number 0128), it is clamped by that value.

4.15 Incremental Designation

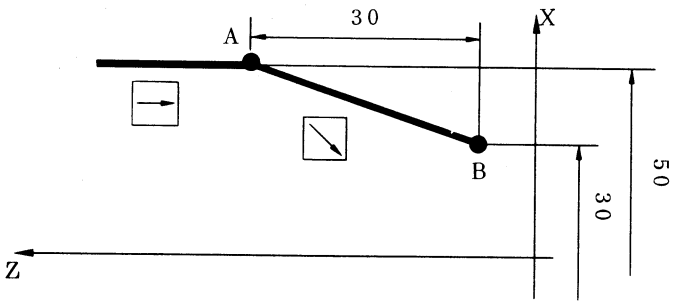
The incremental designation becomes effective for part figure data input such as the end point of a figure element and the center of a circular arc.

The start point of the figure element must be defined in any case.

(1) Incremental designation to a straight line

When the end point of a straight line having a definite start point is input by the incremental value, you have only to input an increment without any need of taking its sign into consideration, since the system judges the arrow direction.

(Example)



End point D X = 1 0 I
Z = 3 0 I

or

End point D X = 3 0
Z = 3 0 I

Assume that a lower rightward arrow was inserted at point A in the above figure.

End point B is located in the minus (−) direction with reference to the start point in both X and Z directions.

However, it is not necessary to attach minus sign (−) as an incremental value.

Also, the radius value is input to X.

(Note) It is rather troublesome to input an X value by the incremental value in a case as shown in the above figure.

In such a case, the Z value only can be input by the incremental value.

(2) Incremental designation to a circular arc

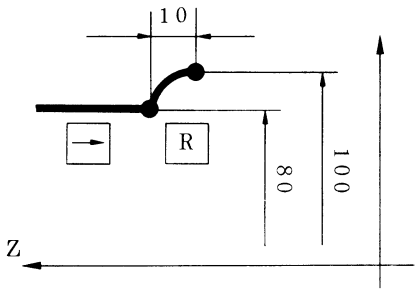
The incremental value can be specified for inputting the end point and center of the circular arc.

Unlike in a straight line, the end point direction as viewed from the start point changes according to the length of circular arc, and its direction cannot be determined automatically.

Accordingly, a signed numerical value must be input.

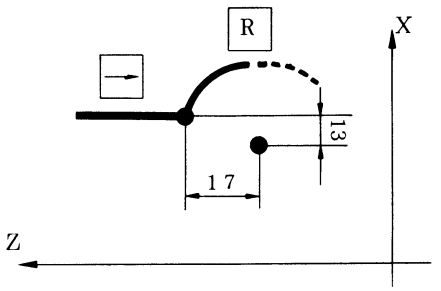
The sign is taken into consideration in the defined coordinate system.

(Example)



END POINT DX = 10 I or 100
Z = −10 I

(Example)



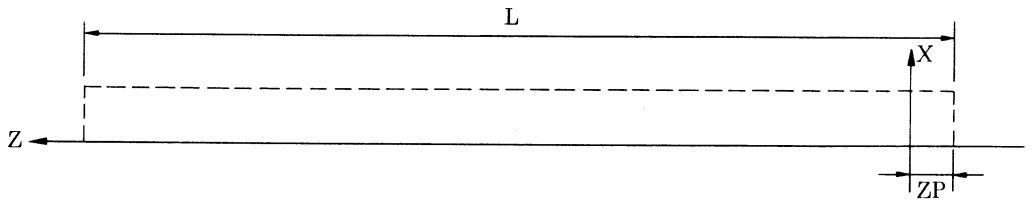
CENTER CDX = −13 I
CZ = −17 I

A signed numerical value must be input, unlike in the incremental designation to a straight line.

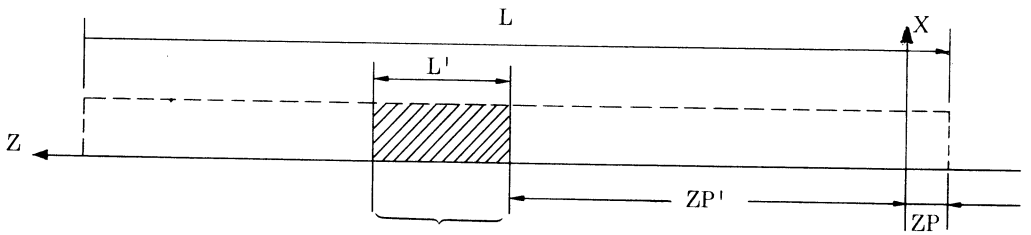
4.16 Partial Expansion of Part Figure

Partial expansion of the input figure can be done by the following method, if a blank is long.

(1) Input all part figures after inputting actual blank dimensions. If the blank is long, the longer part (length in the Z axis direction in the following figure) of the blank is automatically scaled so that it is fully displayed over the entire CRT screen.
As a result, the length in the X-axis direction is displayed smaller as the blank becomes longer than its thickness.



(2) Load the partial blank dimensions again after loading the part figure. The memory stores the previously loaded figure of the entire blank to be machined, and the input blank dimensions are displayed over the entire CRT screen by newly inputting the blank dimensions only. Thus, the part figure is partially expanded and displayed.



This portion is displayed over the entire CRT screen.

If you want to display the shadowed portion illustrated above fully over the entire CRT screen,

Blank dimension L . . . . . Change to width L' to be expanded.

Base line position ZP. . . . . Input ZP' in the above figure as a negative value.

(3) After confirming the figure, reset L' and ZP' to actual figure dimension L and base line position ZP without fail.

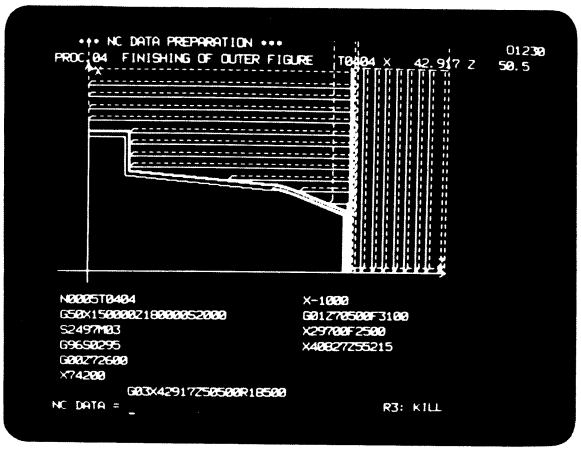
4.17 Single Step

If the F7 key is turned on when producing NC data, processing is stopped before outputting one block of NC data.

The contents of the block to be output are displayed in the second line as viewed from the bottom of CRT screen.

Also, the tool path is displayed when the block concerns the movement.

The following CRT screen shows the stopping state in the single step. Data"G03X42917Z50500R18500"is going to be output.



In this case, the following operation can be done.

(1) If the [SP] key only is depressed, data being displayed in the second line as viewed from the bottom of CRT screen is output, and the Symbolic FAPT returns to the execution to be ready for the next block.

If this [SP] key only is not depressed, that is to say, if the next steps (2) and (3) are made, the execution of Symbolic FAPT does not proceed forward.

(2) NC block can be output from the keyboard by optional blocks under this condition. When the [SP] key only is depressed finally, the display data in the second line as viewed from the bottom of CRT screen is output.

(3) When [DEL] and [SP] keys are input, displayed data output can be deleted.

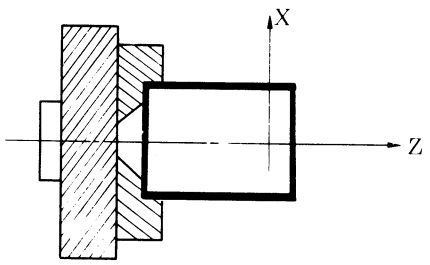
(4) Input [;] and [SP] keys for outputting EOB only.

As a result, a space andEOB code are output.

4.18 Others

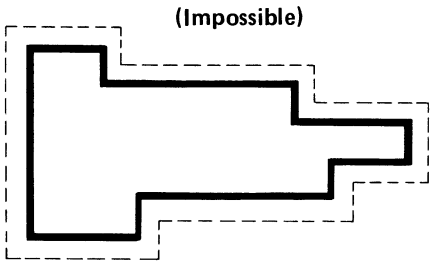
(1) Chucking position

In the Symbolic FAPT, the left side of a blank is chucked as a precondition.

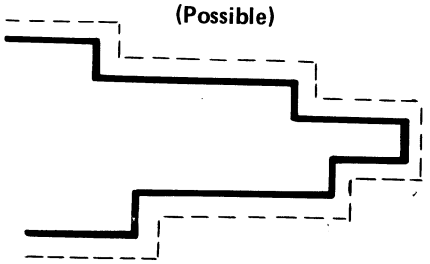


(2) Closed part

No closed part can be defined in case of a special blank.



A figure as shown here cannot be defined.



Define a figure as shown in the left figure.

(3) Change of index position

The machining start points of respective tools are automatically calculated by the index position (DXI, ZI) and setting positions of the tool (XS, ZS).

This automatic calculation is done just after the setting position (XS, ZS) has been input only.

Accordingly, the machining start position cannot be calculated again by changing the index position (DXI, ZI) after it has been automatically calculated once.

When the index position was changed, the setting positions of all tools (XS, ZS) must be input again at the tool data input.

Input the same setting position (XS, ZS) as before again for the purpose of automatically calculating the machining start point to meet the changed index position, even if the setting position (XS, ZS) value remains unchanged when the index position (DXI, ZI) only was changed.

IV. PROGRAMMING USING NC LANGUAGE AND OPERATION AND DISPLAY BY MDI & DPL UNIT

1. NC LANGUAGE

SNC executes programming by Symbolic FAPT function using the graphic display. The programmer has only to depress keys, and there is no particular programming language.

The Symbolic FAPT function produces programs using standard NC language and then, transfers it to the NC memory according to the keying operation of programmer.

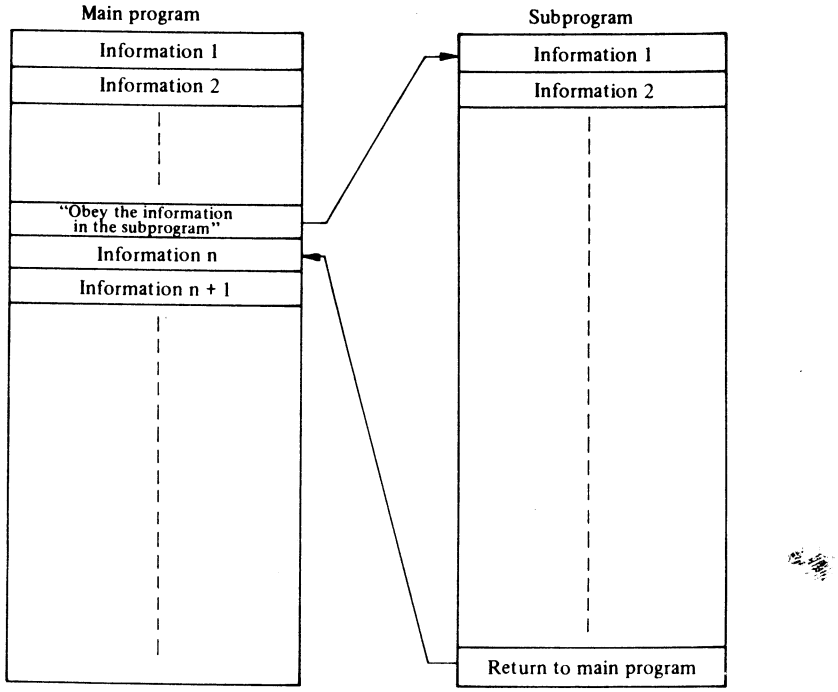
The standard NC language program produced by this Symbolic FAPT is called NC data.

This NC language program has conventionally been employed as an input language to CNC, and called EIA format or ISO format. In FANUC SYSTEM 3T-MODEL D, the NC language in NC memory can be checked, modified, or produced, as required, from MDI & DPL unit. Now, the NC language and the operation using MDI & DPL will be described.

The function items marked with ** in the description of each function in this chapter show the NC language which is not outputted by Symbolic FAPT function.

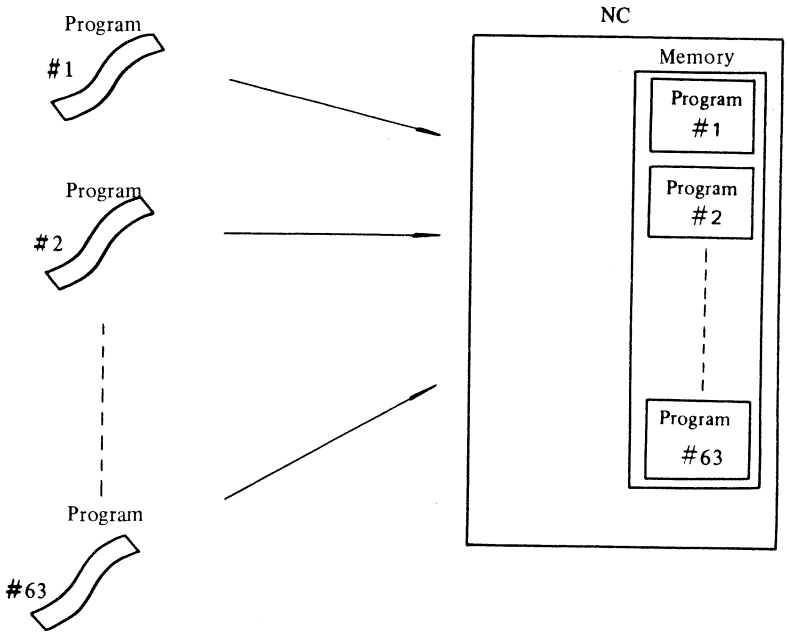
2. COMPOSITION OF PROGRAM

The program is generally divided 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.



A total of 63 main programs and subprograms may be strored in the NC memory.

The NC can move the NC machine tool according to a main program selected from among these main programs.



Refer to section 10.13 for the procedure for storage and selection of one program.

2.1 Block

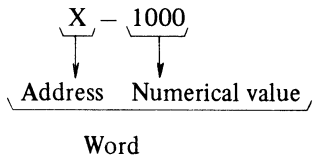
The program is composed of several commands.

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 *.

2.2 Word

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



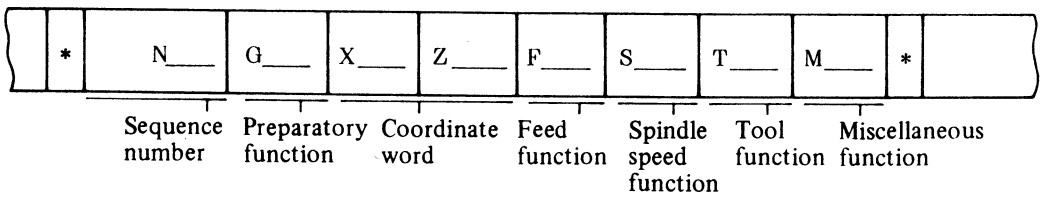
The address is a alphabetic letter which prescribes the meaning of the numerical value following the address.

The addresses and their meanings as used by this system are as follows.

Some addresses may vary their meanings depending on the preparatory functions specified in the program.

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
Feed function	F	Feedrate, thread lead
Spindle speed 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	P	Repetitive count in subprogram

For example, a block is composed of these words as follows:



2.3 Input Format

Each word in a block must be commanded with the fixed format as follows. The input format used by this system 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.

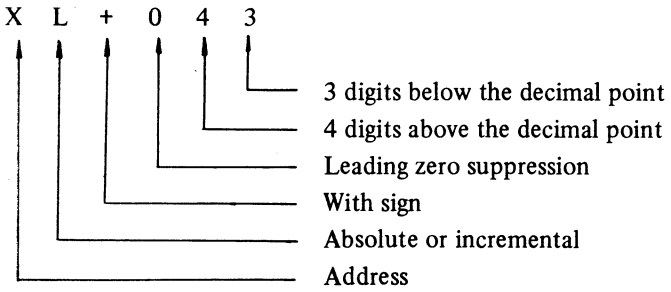
(1) Metric input

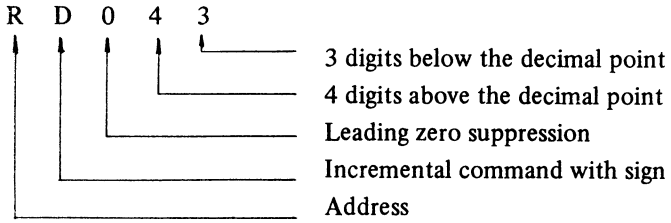
$$N04 \cdot G02 \left\{\begin{matrix}XL+043\\UD043\end{matrix}\right\} \cdot \left\{\begin{matrix}ZL+043\\WD043\end{matrix}\right\} \cdot RD043 \cdot \left\{\begin{matrix}F024\\F040\end{matrix}\right\} \cdot \left\{\begin{matrix}S02\\S04\end{matrix}\right\} \cdot T04 \cdot M02*$$

(2) Inch input

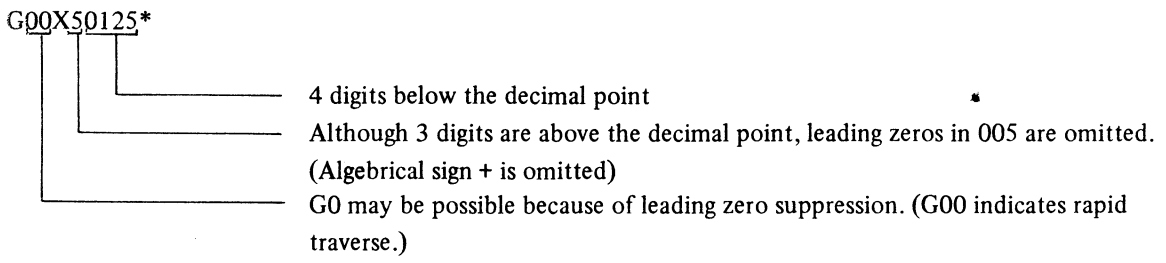
$$N04 \cdot G02 \cdot \left\{\begin{matrix}XL+034\\UD034\end{matrix}\right\} \cdot \left\{\begin{matrix}ZL+034\\WD034\end{matrix}\right\} \cdot RD034 \cdot \left\{\begin{matrix}F016\\F032\end{matrix}\right\} \cdot \left\{\begin{matrix}S02\\S04\end{matrix}\right\} \cdot T04 \cdot M02*$$

(Note 1) Above addresses and numerical values are meant as follows





For example, the command to move a tool to 5.0125 inches on the X axis at the rapid traverse rate is as follows:



(Note 1) The above format omits address P because it has various meanings.

(Note 2) Refer to item 2.4 for decimal point programming.

(Note 3) S04 is optional.

2.4 Decimal Point **

A decimal point may be used with mm, inch or second values.

Z15.0 Z15 millimeters or Z15 inches.

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, R and F

(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) The appropriate G code should be commanded before the numerical values are specified in one block.

(1) G20 (Inch dimension)*

X1.0G04 . . . . Because the value X1.0 is regarded as not the number of seconds but move distances (in inches), X10000G04 is assumed resulting in dwelling for 10 seconds.*

G04X 1.0 . . . G04 X1000 is assumed and it results in dwelling for 1 second.*

(2) G98 (mm/min designation)*

F1.G99 . . . . . Regarded as F1000G99 and 10 mm/rev is assumed. (G99 is mm/rev designation)*

G98 (mm/min designation)*

G99F1. . . . . . Regarded as G99F100 and 1 mm/rev is assumed. (G99 is mm/rev designation)*

(Note 3) There is a great difference between the values with and without the decimal point. Programming differs from electric calculators, etc.

G21 (millimeter dimensions)*

X1. . . . . X1 mm

X1 . . . . . X0.001 mm

G20 (inch dimensions)*

X1. . . . . X1 inch

X1 . . . . . X0.0001 inch

(Note 4) Values with and without a decimal point can be commanded together.

*X1000Z23.7**

*X10.Z22359**

(Note 5) Values less than the least input increment are deleted.

When X1.23456 is commanded, X1.234 is assumed in millimeter input or X1.2345 is assumed in inch input.

•In incremental dimensions, the errors are accumulated. In absolute dimensions, the errors are not accumulated but the error deleted exists at the absolute programming.

Also, the number of digits must not exceed the maximum number of digits allowed.

X1.2345678 . . This is an error because it exceeds 7 digits.

X1.234567 . . . This is not an error because it is within 7 digits.

(Note 6) The number with a decimal point is converted into integer value according to the least input increment in input.

Example

X12.34 ——→ X123400 (input in inch)

This converted integer is checked on its number of digit.

Example

X123456.7 ——→ X1234567000 (input in inch)

An alarm occurs because the converted integer exceeds 7 digits.

2.5 Maximum Programmable Dimensions

The maximum programmable dimensions of each address are listed in table 2.5. Note that these figures give the maximum numerical limit, not the mechanical limit in the NC machine tool. As an NC, the tool may traverse up to 100m (in millimeter input) along X axis, but actually obtainable travelling distance may be limited to 2m on the specific machine tool. Similarly, the NC may permit a maximum cutting feed of 15 m/min, but the NC machine may limit the tool of feed to 6 m/min. When programming, the manual issued by the machine tool builder should be closely consulted, in addition to this manual, to ensure proper programming which will not exceed the actual limitations of your machine tool.

Table 2.5 Basic Addresses and Commandable Range

Function	Address	Input in mm Output in mm	Input in inch Output in mm	Input in mm Output in inch	Input in inch Output in inch
Program number	: (ISO) O (EIA)	1 ~ 9999	Same as left	Same as left	Same as left
Sequence number	N	1 ~ 9999	Same as left	Same as left	Same as left
Preparatory function	G	0 ~ 99	Same as left	Same as left	Same as left
Coordinate word	X, Z, U, W, R	±9999.999 mm	±999.9999 inch	±9999.999 mm	±999.9999 inch
Feed per minute	F	1 ~ 15000 mm/min	0.01 ~ 600.00 inch/min	1 ~ 15000 mm/min	0.01 ~ 600.00 inch/min
Feed per revolution Thread lead	F	0.0001 ~ 500.0000 mm/rev	0.000001 ~ 9.999999 inch/rev	0.0001 ~ 500.0000 mm/rev	0.000001 ~ 9.999999 inch/rev
Spindle speed function	S	0 ~ 9999	Same as left	Same as left	Same as left
Tool function	T	0 ~ 9916	Same as left	Same as left	Same as left
Miscellaneous function	M	0 ~ 99	Same as left	Same as left	Same as left
Dwell	X, U, P	0 ~ 9999.999 sec	Same as left	Same as left	Same as left
Designation of sequence number, repeatitive count	P, Q	1 ~ 9999999	Same as left	Same as left	Same as left

(Note 1) Feed per revolution and thread lead are actually determined by the speed converted into feed per minute with relation to the spindle speed.

The command may be input within the value in the table above.

2.6 Program Number

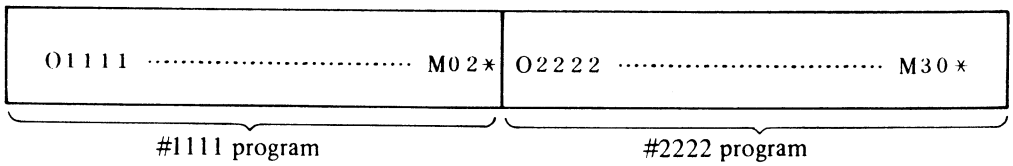
Some program prepared with NC language can be stored in the NC memory. The program number is used to differentiate one program from another.

The program number is identified as follows:

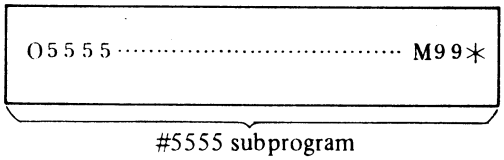
O □□□□

4-digit number (1 ~ 9999, zero should not be used because of compatibility to other NC system. Leading zero suppression.)

A program begins with a program number and finishes with M02*, M30* or M99*,



M99* means the end of a subprogram.



(Note 1) A block with an optional block skip code such as /M02, /M30* or M99* is not regarded as the end of a program.*

(Note 2) When the program number does not exist at the start of the program, the first sequence numbers (N . . .) in that program is regarded as the program number of the program.

(Note 3) When a tape contains more than one program, an EOB code is unnecessary before the second and subsequent programs. However, when a preceding program ends with ER (EIA code) or % (ISO code), an EOB code is necessary at the head of a program. This is because of label skip.

2.7 Sequence Number

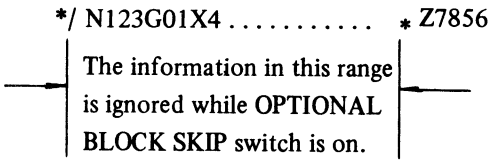
A sequence number can be specified with up to 4-digit number (1 ~ 9999) following address N at the head of a block. The order of sequence numbers is arbitrary and need not be consecutive. Also sequence numbers can be specified in all blocks or in the blocks in which it is required.

It is recommended that sequence numbers should 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.

(Note 1) Sequence number N0 should not be used for compatibility with other NC equipment.

2.8 Optional Block Skip **

When a block has a slash (/) code at the start and the OPTIONAL BLOCK SKIP switch on the operator's panel is turned on, all information in the block with the slash (/) code is ignored during memory operation. If the OPTIONAL BLOCK SKIP switch is off, information in the block with the slash (/) code is effective. That is, the block with a slash (/) code can selectively be skipped depending on the operator's decision.



(Note 1) A slash (/) code must be put at the start of a block. If it is palced elesewhere in the block, the information from the slash (/) code to the character before EOB code is ignored.

(Note 2) The optional block skip is identified when the information is read into the buffer storage from the memory. Even if the optional block skip switch is turned on after the information is read into the buffer, the information which has been already read is not ignored.

(Note 3) This function is also effective during sequence number search.

(Note 4) When storing the program into memory, this function is ineffective. The block with slash (/) code is also stored in the memory irrespective of OPTIONAL BLOCK SKIP switch.

(Note 5) When punching out the program from the memory, the program is punched out irrespective of the OPTIONAL BLOCK SKIP switch.

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. (Refer to item 3.7 absolute and incremental programming.)

Address of coordinate word		Meaning
Basic axes	X Z	Specifies a certain target position in a coordinate system (Absolute programming).
	U W	Specifies a move distance (Incremental programming). U is for X axis and W is for Z axis.
Parameters in circular interpolation	R	Specifies the radius of an arc.

(Example) U-200.00 The tool moves, in the X axis in the negative direction by the distance 200.0.

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 number of controllable axes is two axes, X and Z.

The number of controlled axes in a block is also two axes.

Controlled axes	Simultaneously controlled axes
X, Z	Two

However, in manual operation, the number of simultaneously controlled axes is one.

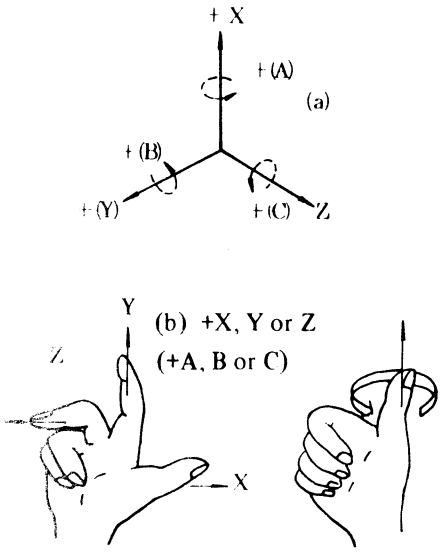
3.1.1 Coordinate axes and symbol of motion

If machine tools provide different correlations between the coordinate axes of the machine tool and the symbols of tool motion, severe confusion will result when programming. Basic concepts on this matter are established in EIA RS-267-A and ISO 841.

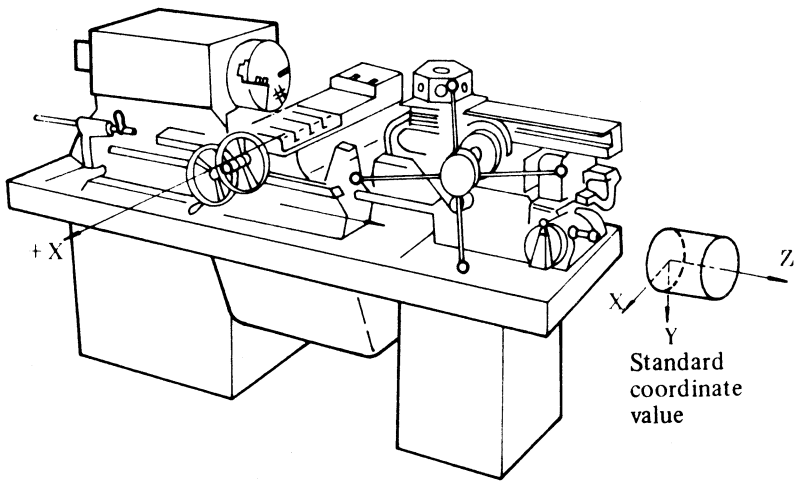
However, care must be exercised on the following points of your programming:

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

(b) When programming, assume that the workpiece stands still and the tool is moved around the workpiece.



The right-hand cartesian coordinate system



Coordinate axes of a turret lathe

3.2 Increment System

The increment system is determined by the following two factors:

3.2.1 Least input increment (Input unit)

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

3.2.2 Least command increment (Output unit)

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

Input and output	Least input increment	Least command increment
Metric input Metric output	X: 0.001 mm (diameter designation) Z: 0.001 mm	X: 0.0005 mm Z: 0.001 mm
	X: 0.001 mm (radius designation) Z: 0.001 mm	X: 0.001 mm Z: 0.001 mm
Inch input Metric output	X: 0.0001 inch (diameter designation) Z: 0.0001 inch	X: 0.0005 mm Z: 0.001 mm
	X: 0.0001 inch (radius designation) Z: 0.0001 inch	X: 0.001 mm Z: 0.001 mm
Metric input Inch output	X: 0.001 mm (diameter designation) Z: 0.001 mm	X: 0.00005 inch Z: 0.0001 inch
	X: 0.001 mm (radius designation) Z: 0.001 mm	X: 0.0001 inch Z: 0.0001 inch
Inch input Inch output	X: 0.0001 inch (diameter designation) Z: 0.0001 inch	X: 0.00005 inch Z: 0.0001 inch
	X: 0.0001 inch (radius designation) Z: 0.0001 inch	X: 0.0001 inch Z: 0.0001 inch

The optional radius designation for X axis should be selected, for radius programming in X axis.

Whether the least command increment system is metric or inch is determined by the individual machine tool and must be selected by setting a parameter (SCW) in advance.

Selection of least input increment between 0.001 mm and 0.0001 inch can be made by G codes or the setting parameter INCH.

G20 ..... Least input increment 0.0001 inch
G21 ..... Least input increment 0.001 mm

Either G20 or G21 which has been selected before the power is turned on is effected at the power on.

3.3 Maximum Stroke

The maximum stroke commandable in this NC is listed in table below:

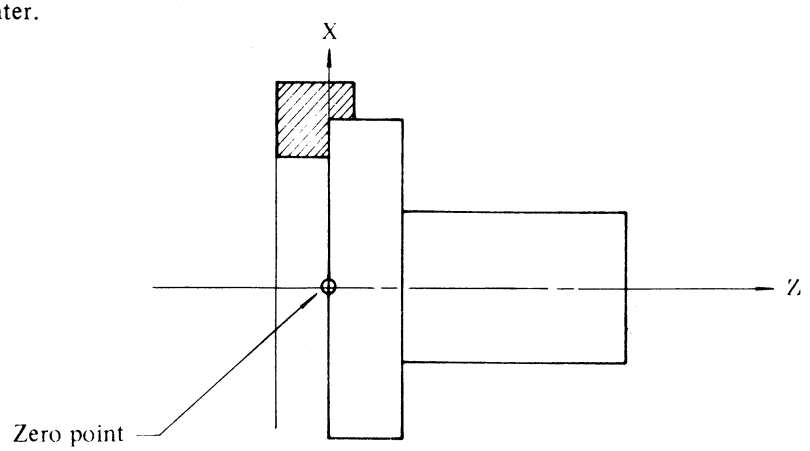
mm input mm output	inch input mm output	mm input inch output	inch input inch output
±9999.999 mm	±999.9999 inches	±9999.999 mm	±999.9999 inches

(Note) Above stroke, of course, varies depending on the machine tool.

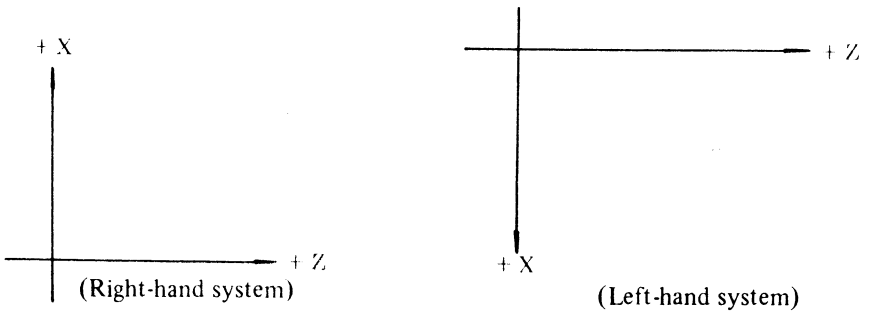
3.4 Program Zero Point and Coordinate System

When programming, a program zero point and a coordinate system must be determined. Usually, program zero point is placed at an arbitrary position on a workpiece.

For example, X axis should be coincided with the workpiece's left face and Z axis should be coincided with the workpiece center.

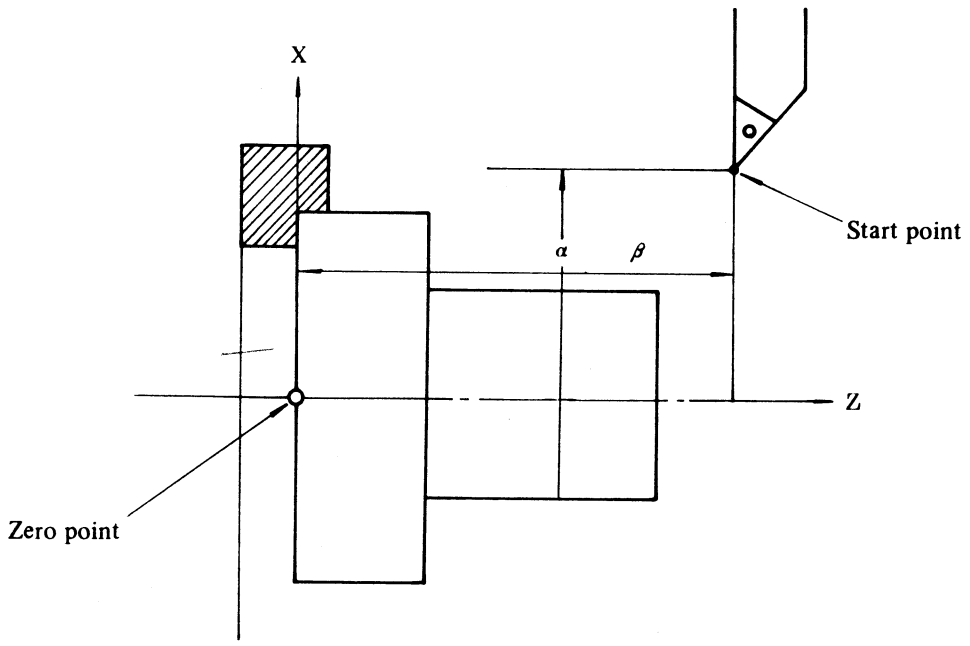


This coordinate system is called work coordinate system. There are two coordinate system. This manual uses the right-hand system.



3.5 Coordinate System and Start Point

The work coorinate system used in programming must be stored into the NC. A tool moves from the start point and the program also starts from the start point. But the NC must know the coordinate value of a tool at the start point by using a G50(G92) command.



A G50(G92) X $\underline{\alpha}$ Z $\underline{\beta}$ * command teaches the NC the work coordinate system.

The G50 command should be programmed at the beginning or commanded by MDI operation after positioning the tool at the start point as shown in the above figure. However, in this case, positioning to the start point must be performed correctly.

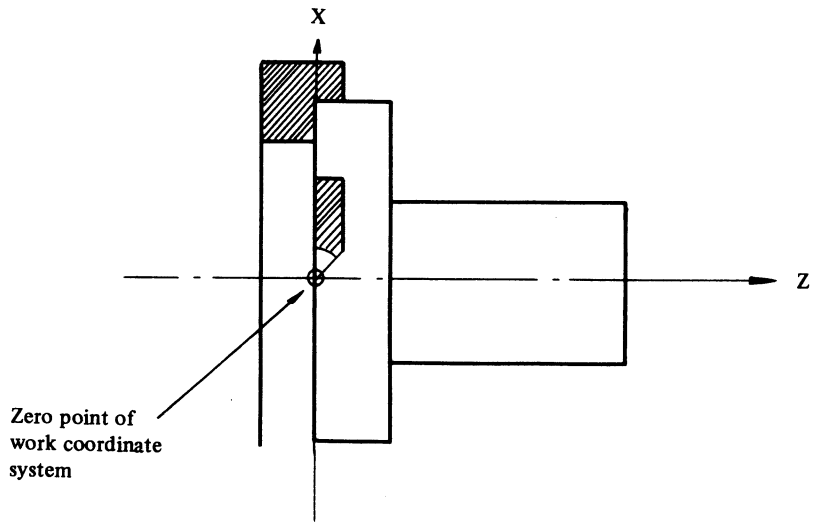
If the work coordinate system is determined at first, the positioning to the start point is performed easily by the following operations.

(1) Positioning with G00 command by MDI operation.

(2) Manual feed using position indicator.

Here, the work coordinate system can be determined by the following method.

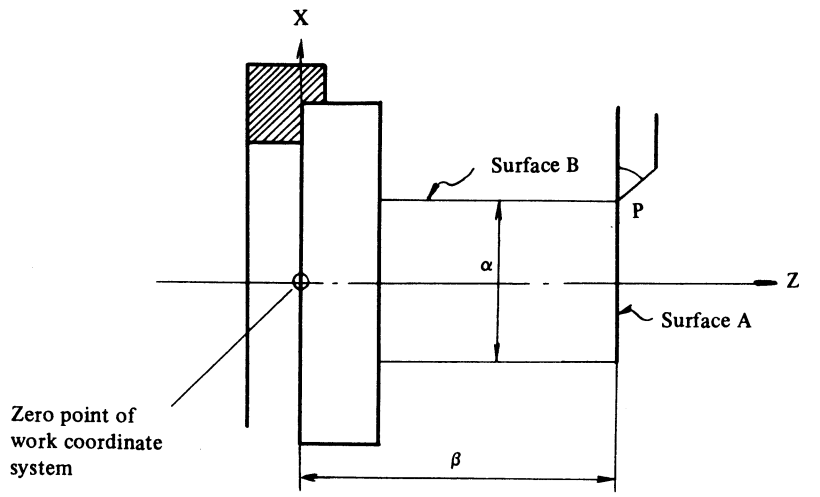
(Method 1) G50 command by MDI operation



If the tool can be positioned at the zero point of the work coordinate system, command following G50 command by MDI operation after positioning.

G50 X 0 Z0 *

However, positioning at the zero point is impossible usually.



At first, position the tool at the point of the workpiece as shown in the above figure (point P). And command the G50 command as follows.

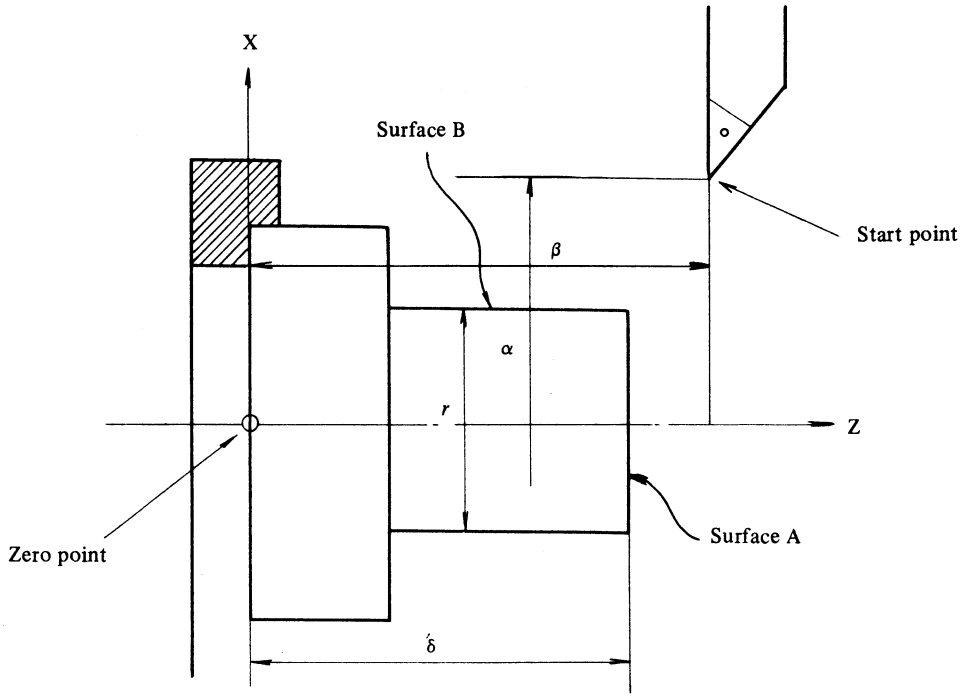
G50 (G92) X α Z β *

Actually, follow the following procedure.

(1) Cut the workpiece along the surface A in manual operation.
(2) Release the tool along X axis without Z axis movement and stop the spindle rotation.
(3) Measure the distance "β" and command G50Z β [INPUT] [START].
(4) Cut the workpiece along the surface B in manual operation.
(5) Release the tool along Z axis without X axis movement and stop the spindle rotation.
(6) Measure the distance "α" and command G50X α [INPUT] [START] and the work coordinate system is set into the NC.

In actual machining, if the tool is exchanged, the offset amount, the difference between the tool in coordinate system setting and tool in actual machining, should be stored in the offset memory according to the machining program.

(Method 2) Coordinate system setting after positioning to the start point.



(1) Cut the workpiece along the surface A in manual operation.
(2) Release the tool along X axis without Z axis movement and stop the spindle rotation.
(3) Reset the relative coordinate value of Z axis (address W) (refer to item IV. 4.10).
(4) Measure the distance "δ" in the above figure.
(5) Cut the workpiece along the surface B in manual operation.
(6) Release the tool along Z axis without X axis movement and stop the spindle rotation.
(7) Reset the relative coordinate value of X axis (address U) (refer to item IV. 4.10).
(8) Measure the distance "γ".
(9) Position the tool at the point with the following coordinate values.

$$x = \alpha - \gamma$$
$$z = \beta - \delta$$

Here, α and β is the coordinate values of the start point.

(If the tool is exchanged in actual machining, set the offset amount as in method 1)

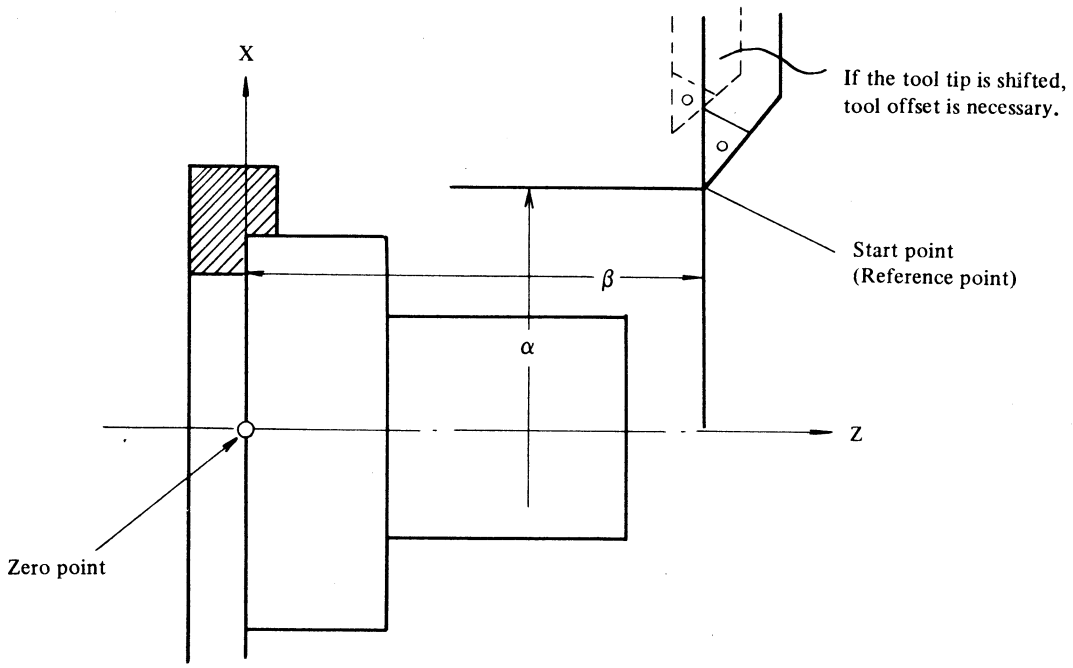
And if G50 (G92) command as follows is programmed at the beginning after that, the desired work coordinate system is set in program execution.

G50 (G92) X $\underline{\alpha}$ Z $\underline{\beta}$ *

(Method 3) Use the reference point as a start point.

Positioning at the start point is performed by the reference point return. And then follow the method 2. However, the tool tip may not be positioned at the reference point. Accordingly, the tool standard point is positioned at the reference point.

The difference between them should be compensated by tool offset function in item III. 6.1 or work coordinate system shift in item III. 5.7.



(Method 4) Automatic coordinate system setting.

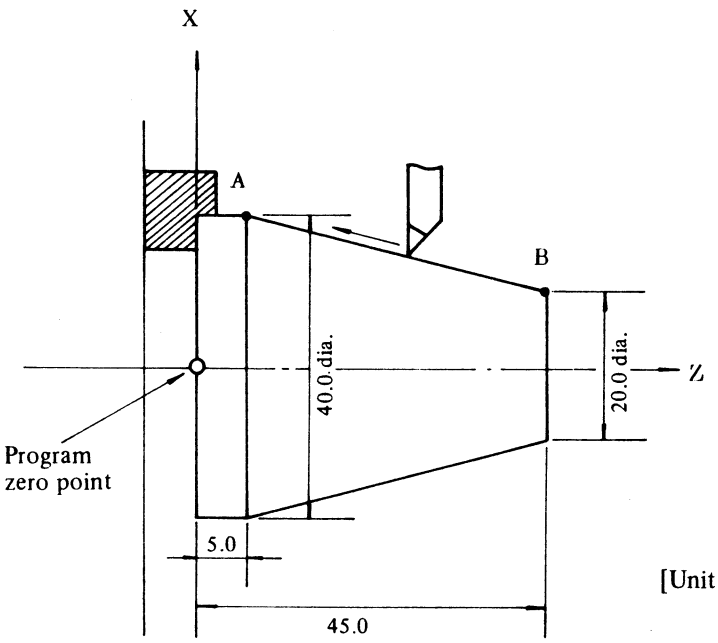
If the corresponding parameter is set, the work coordinate system is set in manual reference point return. Set the coordinate values α and β to the parameters with parameter number 76 and 77 respectively. The coordinate system, in which the reference point coordinate values are α and β, is set. As the tool tip may not be positioned at the reference point as described in method 3, the tool position should be compensated by tool offset function or work coordinate system shift function.

3.6 Reference Point

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

Accordingly, a program may not start from a certain position in the work coordinate system but may start from a reference point. In this case, because the reference 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 reference point must be commanded by a G50(G92) command in the work coordinate system.

3.7 Absolute and Incremental Programming



[Unit: inch]

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)	X40.0Z5.0*
Incremental programming	Specifies a distance from start point to end point	U (Distance along X axis) W (Distance along Z axis)	U20.0W-40.0*

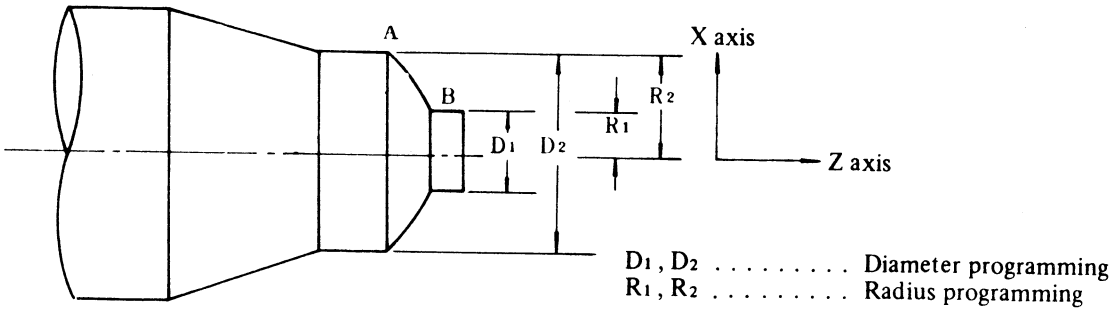
(Note 1) Absolute and Incremental commands can be used together in a block. In the above example, a command as follows is possible.

*X40.0W-40.0**

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

3.8 Diameter and Radius Programming **

Since the workpiece cross section is usually circular in NC lathe control programming, its dimension is specified in two ways, diameter and radius programming.



When the diameter is specified, it is called diameter programming and when the radius is specified, it is called radius programming. If a radius programming option is equipped, the radius programming can be performed. 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. In the above figure, specifies from D2 to D1 for tool path B to A.
Coordinate system setting (G50/G92)	Specifies a X axis coordinate value with a diameter.
X component of tool offset value	Parameter setting decides either diameter or radius value.
Parameters in G90, G92, G94 such as cutting depth along X axis. (R)	Commands radius value
Radius designation in circular interpolation (R)	Commands radius value
Feedrate along X axis	Change of radius/rev Change of radius/min
Display of X-axis position	Displayed in diameter value.

(Note 1) In the following explanations, although diameter or radius programming is not specified, a X axis graduation is a diameter value when considering diameter programming and it is a radius value when considering radius programming.

(Note 2) The meaning of using a diameter value for the tool offset value is that when the outer diameter is cut with the tool offset value changed, outer diameter changes in accordance with a diameter value. When offset value changes by 10 mm with the tool unchanged, the outer diameter changes by 10 mm in its diameter value.

(Note 3) The meaning of using a radius value for tool offset value is that tool length itself can be set.

4. FEED FUNCTION

4.1 Rapid Traverse Rate

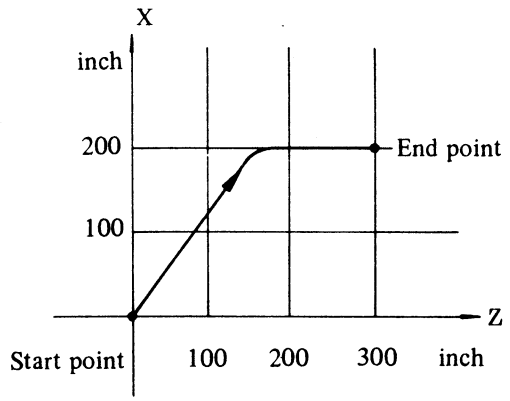
At rapid traverse, the machine moves at the specified rapid traverse rate of each axis.

Usually, this rapid traverse rate is determined and set (by parameter) by the machine tool builder before shipping.

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

For example, when X and Z axis rapid traverse rates are 500 inch/min and 800 inch/min respectively and the following command is programmed, G00U200.0W300.0* the times in which the machine moves between start and end point of X and Z axes are 12 sec and 22.5 sec respectively. If radius programming is specified in X axis, the times is 24 sec.

(The tool path in above example)

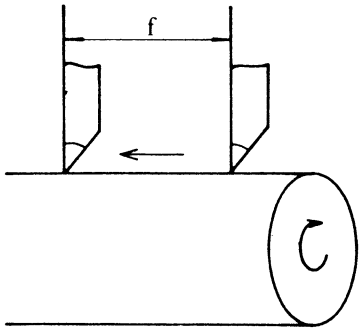


(Diameter programming)

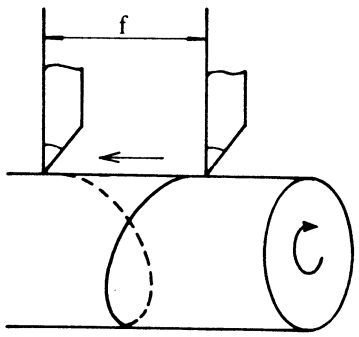
Override can be applied to a rapid traverse rate with the switch on the machine operator's panel. (Fo, 25%, 50% 100%) Fo is specified by parameter setting and its unit are not percent (%), but mm/min or inch/min.

4.2 Cutting Feedrate

Cutting feed rate and tool feed rate can be specified by the number following the address F.
It is specified in feed per revolution or feed per minute.



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



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

		Feed per minute	Feed per revolution
		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 (F1 ~ F15000)	0.0001 ~500.0000mm/ rev (F1 ~ F5000000)
	Input in inch	0.01 ~ 600.00 inch/min (F1 ~ F60000)	0.000001 ~ 9.999999 inch/ rev (F1 ~ F9999999)
Clamp value		Cutting feed rate is clamped at the constant rate in common. This clamp value is specified by the machine tool builder. (If the cutting feed rate attempts to over-ride the clamp value by the override function, it is also clamped)	
Override		Override can be applied 0 to 150% in 10% increment.	

The clamp value is specified in mm/min or inch/min. In feed per revolution, cutting feed rate should be converted in mm/min or inch/min by the following expression.

fm = fr X R

fm: Feed per minute (mm/min or inch/min)

fr: Feed per revolution (mm/rev or inch/rev)

R: Spindle speed (rpm)

(Note 1) 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.

(Note 2) Except in the course of acceleration or deceleration, accuracy of the NC operation for commanded feed rate is retained within ±2% of the commanded rate.

(Note 3) Up to 7 digits are permitted in the F code input. However the feed rate cannot exceed the clamp value.

(Note 4) If the feed rate is programmed in feed per revolution, a position coder must be mounted on the spindle.

(Note 5) If the revolution of the position coder is 1 rpm or less, the cutting feed rate is not uniform. If the nonuniformity of revolution does not affect to the machining, it may be performed. However, the slower the spindle speed becomes, the more nonuniform the cutting feed rate becomes.

4.3 Lead Length of Thread

In thread cutting, the lead length is commanded by the number following addresses F. The thread cutting is commanded by G32 (G33), or G92 (G78).

The range of lead length is as follows:

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

The range of lead length is as follows:

	Input in metric	Input in inch
F	0.0001 ~ 500.0000	0.000001 ~ 9.999999

Spindle speed is limited as follows:

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

(Provided that $R \geqq 1$)

R: Spindle speed (rpm)
Lead: mm or inch
Max. feed rate: mm/min or inch/min

Max. feed rate is the smaller of the following rates;
Max. commanded feed rate in feed per minute and the max. feed rate limited by motor or machine capacity.

(Note 1) The spindle speed is read, by a position coder mounted on the spindle, it is converted into feed per minute and the tool is moved at the converted feed rate.

(Note 2) In thread cutting, the cutting feedrate override is ineffective and it is clamped at 100%.

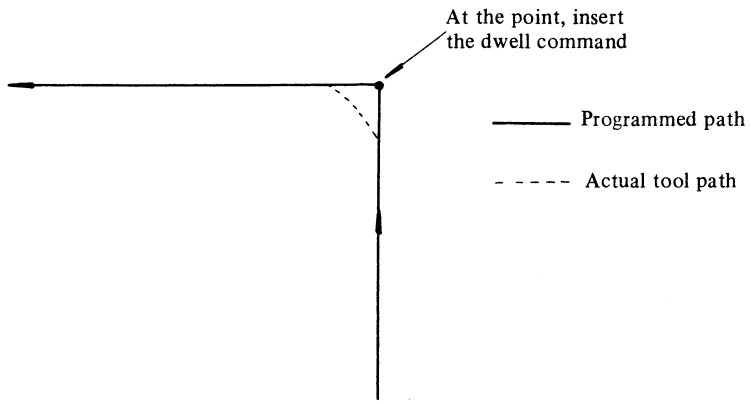
(Note 3) The converted cutting feed rate is clamped.

Address F is used commonly for thread lead, feed per minute and feed per revolution.

4.4 Automatic Acceleration and Deceleration

In feed start or feed stop, the acceleration or deceleration with certain time constant is applied automatically so that the machine system is not jarred. When programming, the above problem is not of concern.

However, because of automatic acceleration and deceleration, a corner is not cut sharply. In this case, insert the dwell command (G04) between the two blocks.



If the dwell command is programmed, the actual tool path coincides with the programmed path. The faster the cutting feed rate becomes, the larger the error at the corner becomes.

(Note 1) The following chart shows the feed rate changes between blocks of information in which there are different types of movement.

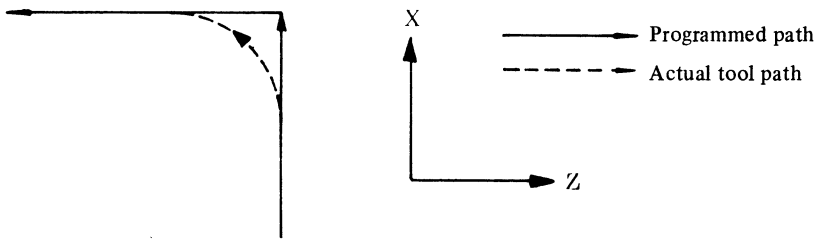
Previous block / *New block*	*Positioning*	*Cutting feed*	*Unmoving*
Positioning	X	X	X
Cutting feed	X	O	X
Unmoving	X	X	X

X : *After the commanded feed rate has decelerated to zero, the next block is executed.*

O : *Next block is executed sequencially so that the commanded feed rate is not changed extremely.*

(Note 2) The actual tool path does not coincide with the programmed path, because acceleration and deceleration are applied in each axis (X and Z) independently and each axis feed rate is changed between blocks.

For example, if the tool moves in the X axis only in one block and in the Z axis in the next block, the feed rate in the X axis is decelerated during acceleration of the Z axis and the actual tool path is as follows.



In circular interpolation, the actual arc radius is smaller than that of the programmed arc. For the error of the above, refer to the appendix.

5. PREPARATORY FUNCTION (G FUNCTION)

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	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.

(Example)

G01 and G00 are modal G codes in 01 group.

G01X —— *
Z —— *
X —— *
G00Z —— *

} G01 is effective in this range.

G code	Group	Function	Classification
G00	01	Positioning	B
G01		Linear interpolation	B
G02		Circular interpolation (CW)	B
G03		Circular interpolation (CCW)	B
G04	00	Dwell	B
G10		Offset value setting	O
G20	06	Inch data input	O
G21		Metric data input	O
G27	00	Reference point return check	B
G28		Return to reference point	B
G32	01	Thread cutting	B

G code	Group	Function	Classification
G50	00	Programming of absolute zero point, setting of max. spindle speed.	B, O
◤ G68	04	X axis mirror image ON	O
G69		Mirror image cancel	B
G70	00	Finishing cycle	B
G71		Stock removal in turning	B
G72		Stock removal in facing	B
G73		Pattern repeating	B
G74		Peck drilling in Z-axis	B
G75		Grooving in X-axis	B
G76		Thread cutting cycle	B
G90	01	Cutting cycle A	B
G92		Thread cutting cycle	B
G94		Cutting cycle B	B
G96	02	Constant surface speed control	O
◤ G97		Constant surface speed control cancel	B
G98	05	Feed per minute	B
◤ G99		Feed per revolution	B

B: Basic O: Option

(Note 1) The G codes marked with ◤ are initial G codes in each group. That is, when turning power on, this G code is set.

(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. (Alarm number 010)

(Note 4) A number of G codes can be commanded in a block if they are 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.

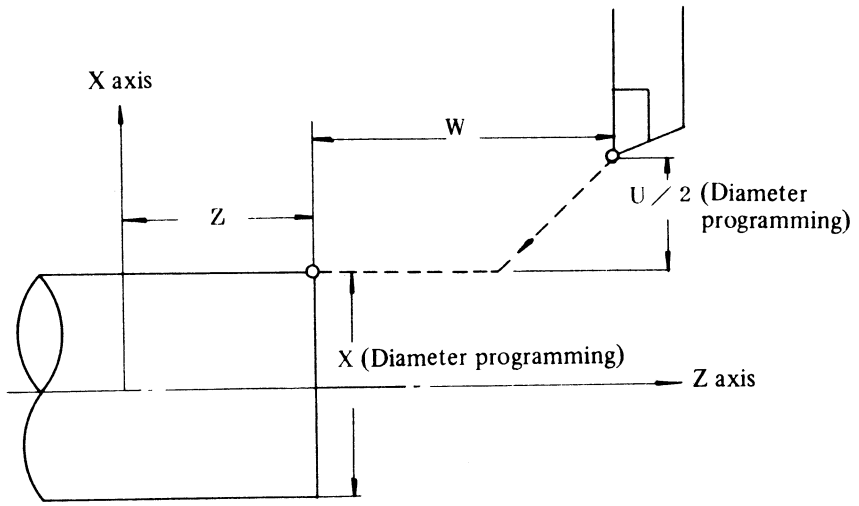
(Note 5) A G code from each group is displayed.

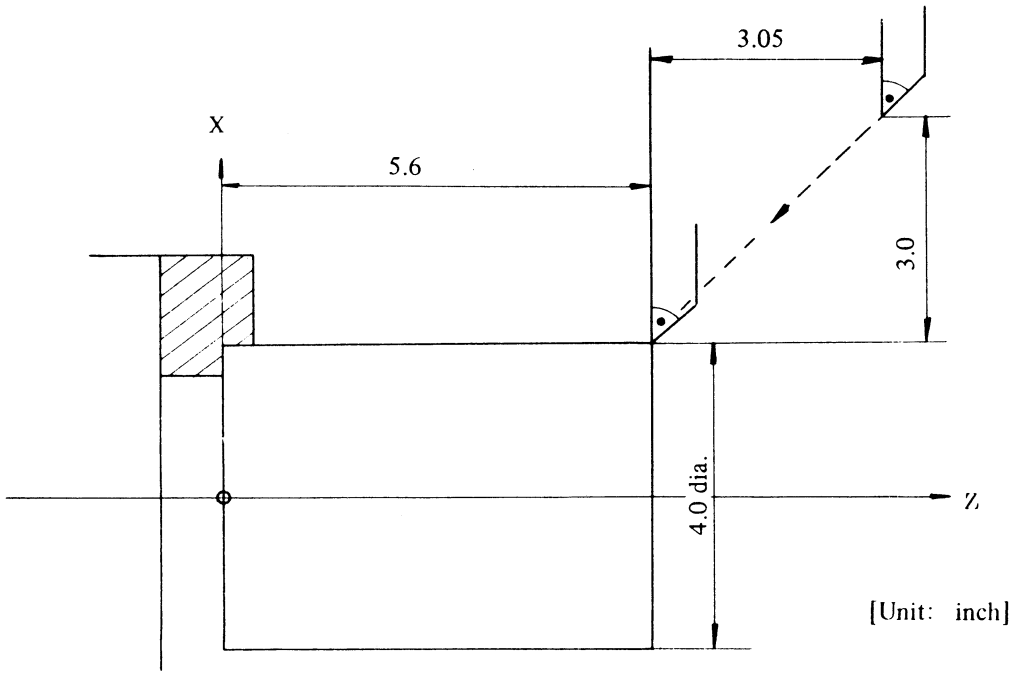
5.1 Positioning G00

G00 specifies positioning.

A tool moves to the (X, Z) position in the work coordinate system or from its current position to the position specified as the (U, W) distance at the rapid traverse rate in each axis independently.

G00X(U) —— Z(W) ——*





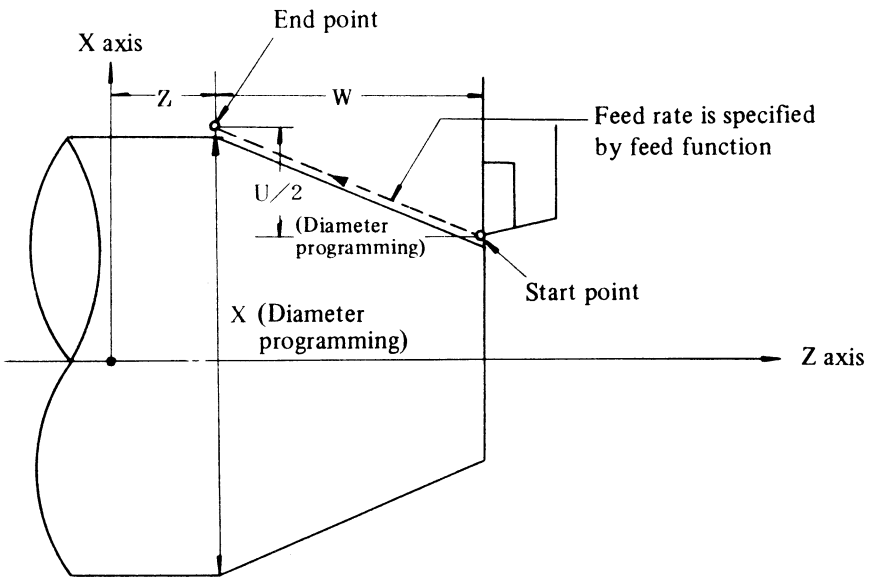
(Radius programming) G00X2.0Z5.6* or G00U-3.0W-3.05*

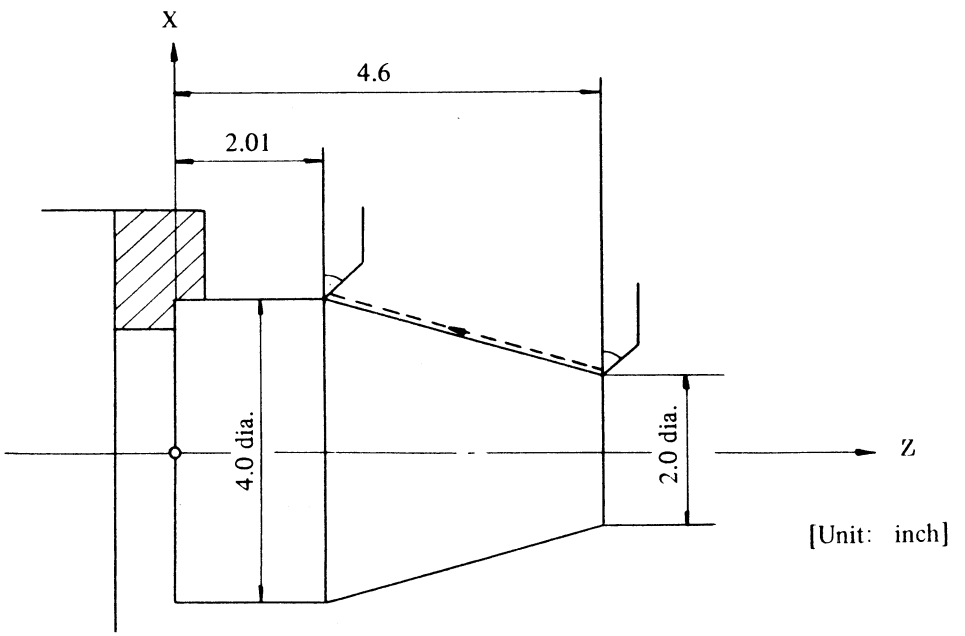
(Note 1) The rapid traverse rate in the G00 command is set for each axis independently by the machine tool builder. Accordingly, rapid traverse rate cannot be specified by the address F.
In the positioning mode actuated by G00, the tool is accelerated to a predetermined speed at the start of the block and is decelerated at the end of the block, and the execution proceeds to the next block.

5.2 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) distance at the feedrate commanded by address F. Linear interpolation is specified by G01.

G01X(U) —— Z(W) —— F ——*





(Diameter programming) G01X4.0Z2.01F2.0* or G01U2.0W-2.59F2.0*

The feedrate commanded by the address F is the speed that the tool moves along the straight line. If the feedrate has not yet been commanded with an address F, it (feedrate) is zero.

(Note 1) Feedrate along each axis is as follows:

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

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

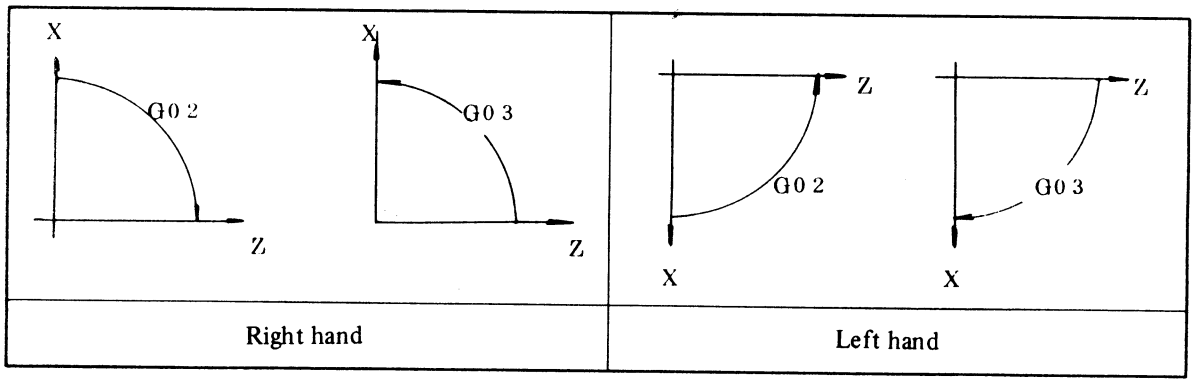
When *F:* *Commanded feedrate*
L: *Move distance*
Lx: *Move distance along X axis*
Lz: *Move distance along Z axis*

5.3 Circular Interpolation G02, G03

The command below will move a tool along a circular arc. The commands are given in the followings format:

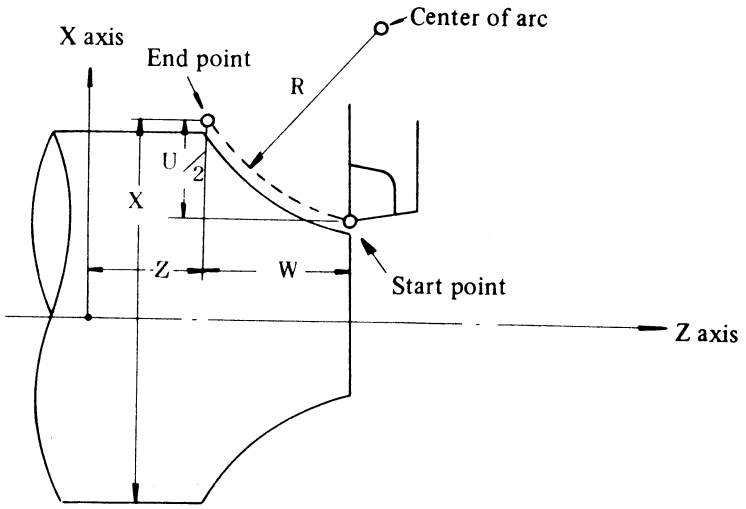
Data to be given		Command	Meaning
1	Rotation direction	G02	Clockwise direction (CW)
		G03	Counterclockwise direction (CCW)
2	End point position	X, Z	End point position (X, Z) in the work coordinate system.
	Distance to the end point	U, W	Distance from start point to end point.
3	Radius of arc center	R	Radius of arc center. However arc is up to 180 degrees. (Always radius value)

Clockwise direction or counterclockwise direction varies with right or left hand coordinate system.

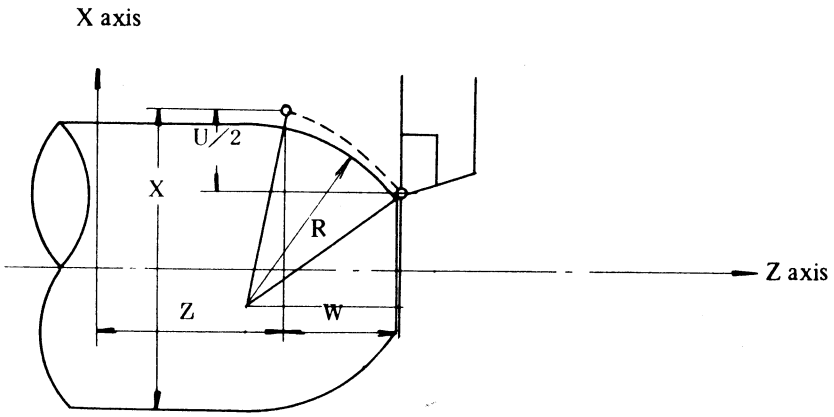


Right hand | Left hand

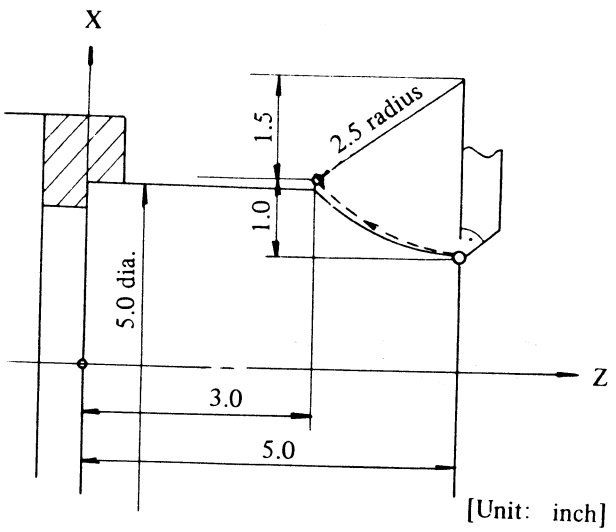
G02X(U) ____ Z(W) ____ R ____ F____ * (Diameter programming)



G03X(U) ____ Z(W) ____ R ____ F ____ * (Diameter programming).



(Diameter programming)
G02X5.0Z3.0R2.5F0.03*
or G02U2.0W-2.0R2.5F0.03*



[Unit: inch]

Feedrate in circular interpolation is specified by F code. Feedrate along an arc (Feedrate tangent to an arc) is controlled to maintain the specified feedrate.

(Note 1) 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.

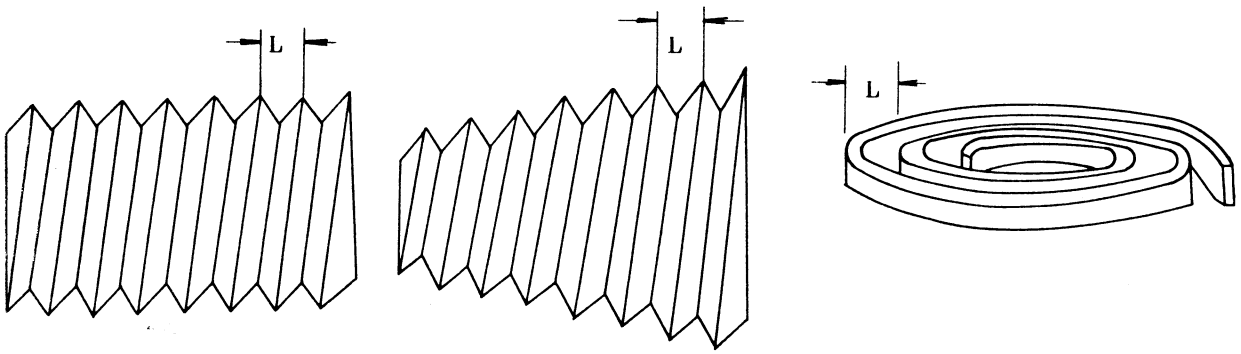
(Note 2) When address R specifies a value less than half of the distance between the start point and the end point, a half circle will be generated.

(Note 3) Feedrate in circular interpolation is specified with an address F. Refer to item 4.2 Feedrate. Error of the feedrate commanded against actual feedrate is with ±2%.

(Note 4) If address R is omitted, the tool moves to the end point linearly.

5.4 Thread Cutting G32

Tapered screws and scroll threads in addition to straight threads can be cut by the use of a G32 (G33) command.

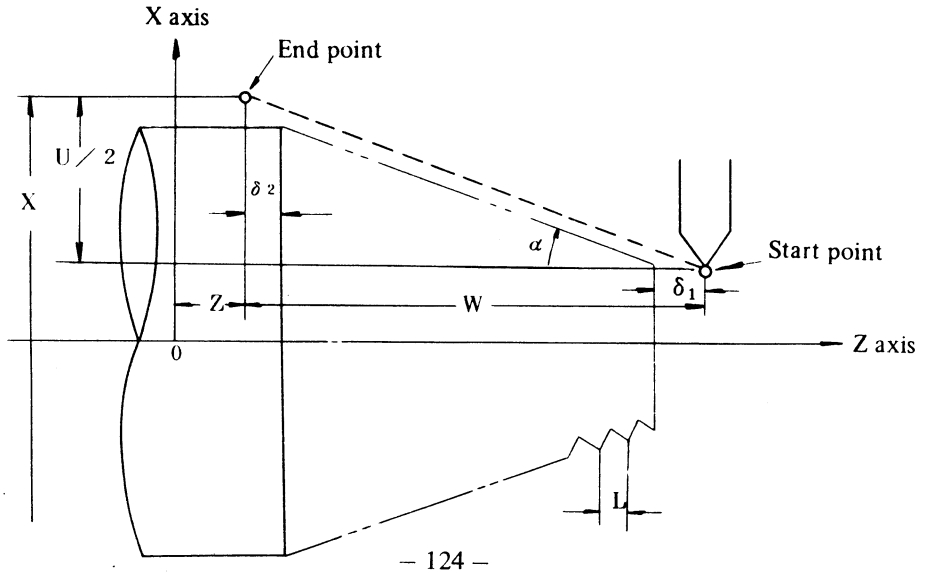


Straight thread　　Tapered screw　　Scroll thread

The command shown below enables thread cutting with the lead specified by numeric values following address F.

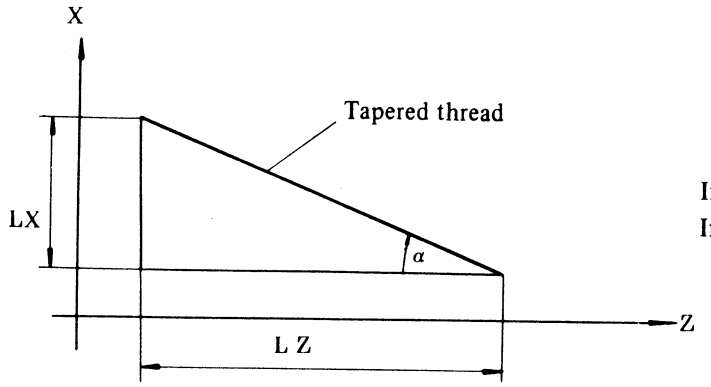
The end point is specified by coordinate position (X, Z) in the work coordinate system or by move distance (U, W) from start point to end point.

G32 X (U) ___ Z(W) ___ F ___ *



In general, thread cutting is repeated on the same tool path, in the course of rough cutting and finish cutting for a screw. Since the thread cutting is started when the position coder mounted on the spindle detects a 1-turn signal, the threading 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 cutting tapered screws, the lead whose value is greater either in the X or Z direction must be commanded.



If $\alpha \leq 45°$, lead is LZ.
If $\alpha > 45°$, lead is LX.

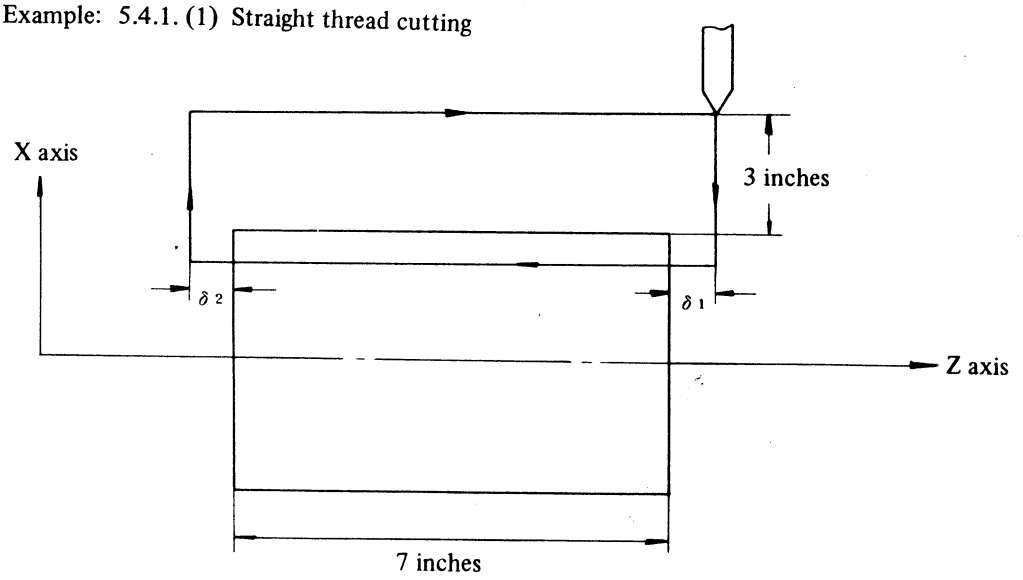
Thread lead must be specified by radius value.

In general, the lag of the servo system, etc. will produce somewhat incorrect leads at the starting and ending points of thread cut. Considering this fact, the threading length should be commanded longer, to some extent, than required thread length. Refer to item 4.3 for thread lead.

Example: Command value and thread lead

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

Example: 5.4.1. (1) Straight thread cutting



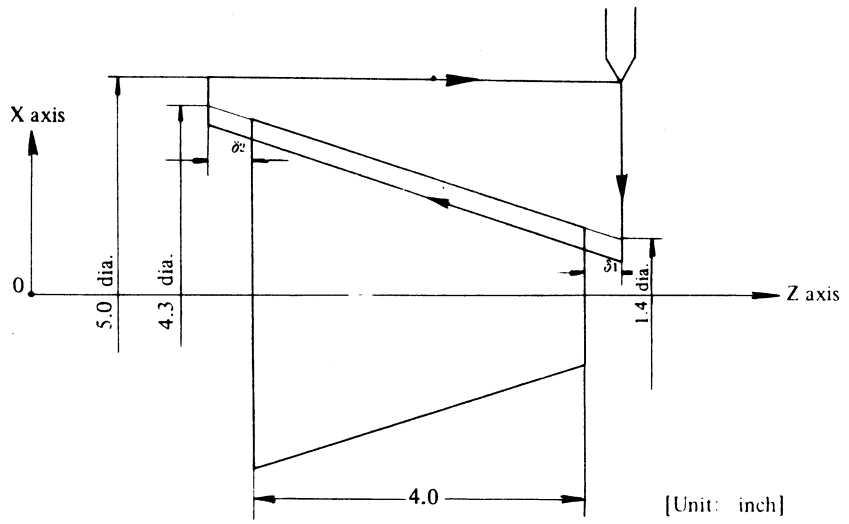
Programming is made with the following values:

Thread lead : 0.4 inch
δ_1 : 0.3 inch
δ_2 : 0.15 inch
Depth of cut : 0.1 inch (Cut twice)

(Inch input, radius programming)

```
G00     U-3.1 *
G32     W-7.45 F0.4*
G00     U 3.1*
        W 7.45*
        U-3.2* (For the second cut, cut 0.1 inch more)
G32     W-7.45*
G00     U3.2*
        W-7.45*
```

Example: 5.4.1.(2) Taper thread cutting



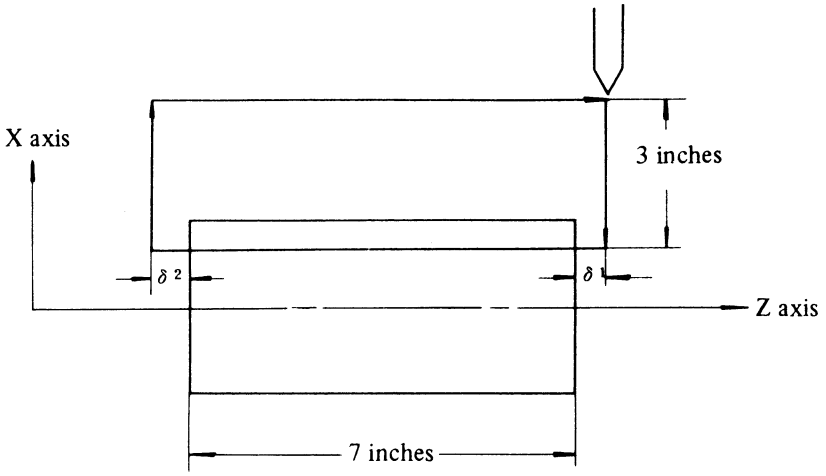
Programming is made with the following values:

Thread lead : 0.35 inches in the direction of Z axis
δ_1 : 0.2 inches
δ_2 : 0.1 inches
Cutting depth in X axis direction is 0.1 inch (Cut twice)

(Inch input, Diameter programming)

```
G00     X 1.2*
G32     X 4.1W-4.3 F0.35*
G00     X 5.0*
        W 4.3*
        X 1.0*
G32     X 3.9W-4.3*
G00     X 5.0*
        W 4.3*
```

Example: 5.4.1. (3) Precision thread cutting



Thread lead : 3 pitch per inch (approx. 0.333333 inch)
δ_1 : 0.3 inch
δ_2 : 0.15 inch
Depth of cut : 0.1 inch (Cut twice)

(Inch input, Radius programming)

G00	U-2.9*
G32	W-7.45F 0.333333*
G00	U 2.9*
	W 7.45*
	U-3.0* (For the second cut, cut 0.1 inches more)
G32	W-7.45*
G00	U 3.0*
	W 7.45*

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

(Note 2) Dry run is also effective during thread cutting.

(Note 3) It is very dangerous to stop feeding the thread cutter without stopping the spindle. It will suddenly increase the cutting depth. Thus, the feed hold function is ineffective while thread cutting. If feed hold is pressed and held during thread cutting, the tool will stop after a block (not for thread cutting) is executed as if SINGLE BLOCK execution.
However, feed hold lamp (SPL lamp) is lighted when the FEED HOLD button on the machine control panel is pushed. Then, when the tool stops, the lamp is turned off (Single Block stop state).

(Note 4) When the FEED HOLD button is again pushed in the first non thread cutting block or when it has been continuously pushed, the tool stops at that time.

(Note 5) When thread cutting is executed in the single block state, the tool stops after the execution of the first non thread cutting block.

(Note 6) When the mode is changed from automatic operation mode to manual operation mode during thread cutting, a tool stops at the first non thread cutting block as when the feed hold button is pushed and held as mentioned in (Note 4).
However, when the mode was changed from one automatic operation mode to another automatic operation mode, the tool stops after execution of non thread cutting block as well as the case in (Note 3) at single block mode.

(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.

*G32Z __ F___ **
*Z __ ** *(An 1-turn signal is not detected before this block.)*
*G32** *(Even this block is regarded as a thread cutting block.)*
*Z __ F ___ ** *(Accordingly, an 1-turn signal is also not detected before this block.)*

(Note 8) Because constant surface speed control is effective during scroll thread or tapered screw cutting and spindle speed changes, correct thread lead may not be cut. Accordingly, constant surface speed control should not be used during thread cutting.

5.5 Automatic Reference Point Return G27, G28 **

5.5.1 G27 Reference point return check

The reference point is a fixed point on the machine and the manual reference point return can move a tool to the reference point.

A G27 command confirms whether a tool has reached the reference point or not by the program which was made to return it to the reference 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 reference point, the reference point return LED lights. If only one axis has reached the reference point, the reference point return LED of that axis lights. If the commanded axis is not on the reference point after executing of reference point return, an alarm will be generated.

(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 reference point, the LED does not light. Normally, offset should be cancelled before a G27 is specified.

(Note 2) In inch machine with metric input, even if a program misses the reference point by 1μm, the reference point return LED may light. This is because the least command increment is smaller than the input unit.

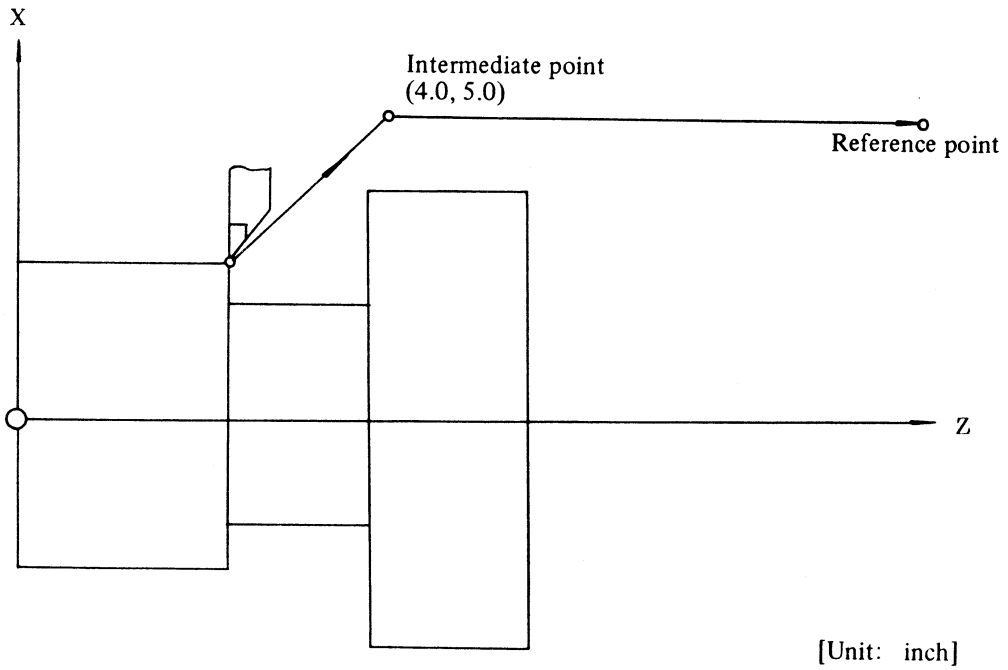
5.5.2 G28 Automatic return to reference point

G28X(U) —— Z (W) —— *

This command provides an automatic return to reference point for commanded axes. Positioning to the reference point is made by first going through the point specified by X—— Z —— *.

The G28 block functions to position all commanded axes to the intermediate point at rapid traverse rate, and, if the machine lock has not been effected, the Reference Point Return LED lights.

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



G28X4.0Z5.0 *

In general, tool offset should be cancelled, in automatic reference point return.

5.6 Dwell G04

G04X (t) * or G04U (t) * or G04P (t) *

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

The maximum command time is 9999.999 seconds. Error of time t is approximately within 16ms.

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 moving velocity of the previous block has become zero.

5.7 Programming of Absolute Zero Point G50

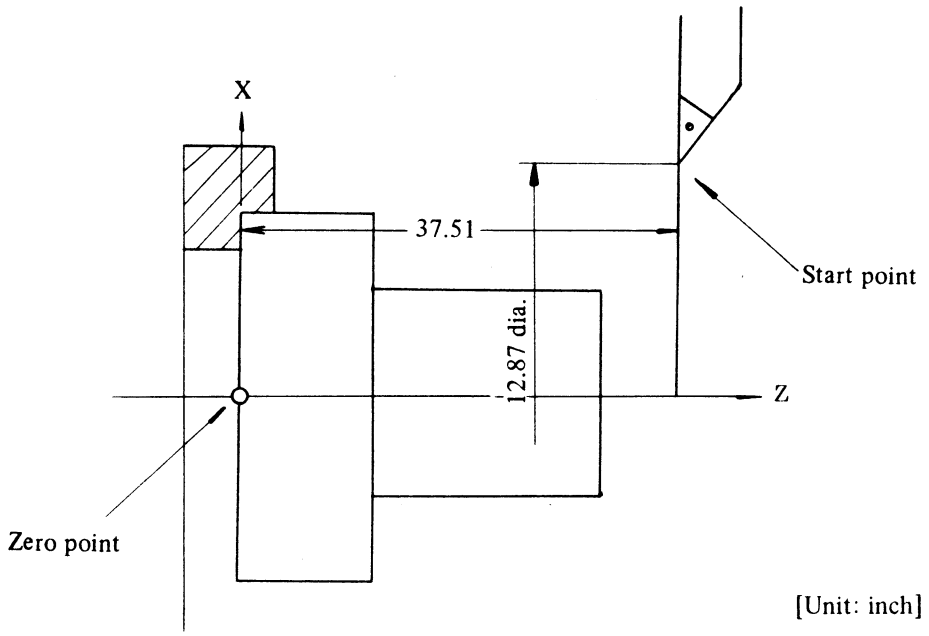
When it is desired to move a tool to a certain position by absolute command, the coordinate system must be set in advance.

5.7.1 The coordinate system is established by the following command;

G50 X (x) Z (z) *

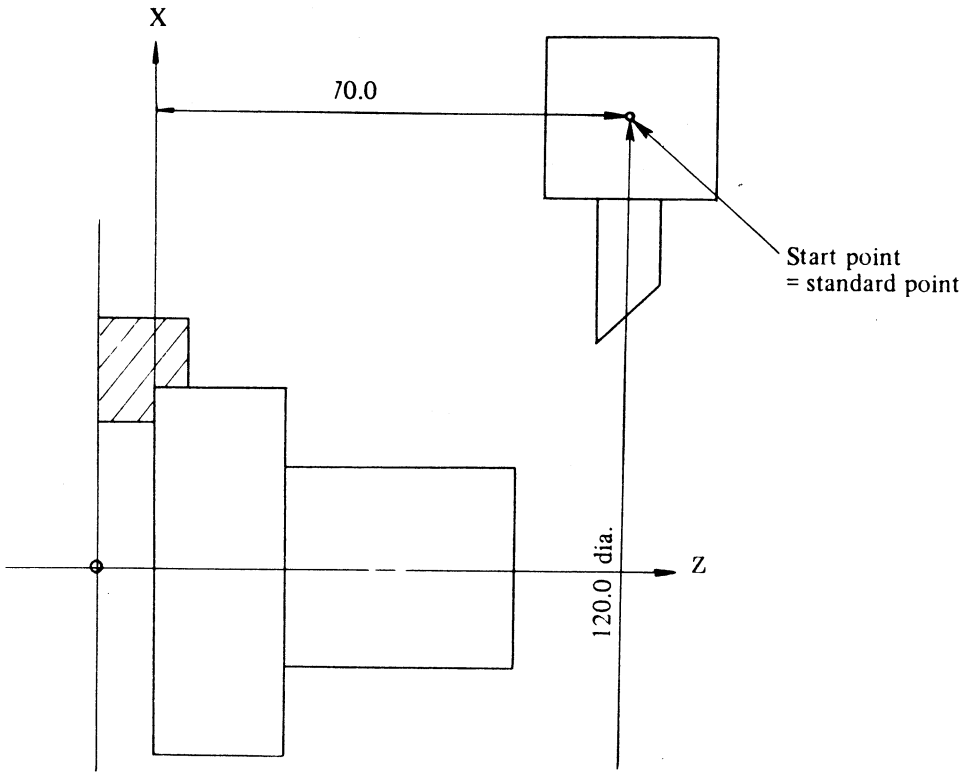
This command creates a coordinate system in which the coordinate values of the tool position, such as tool tip, is x and z. This coordinate system is called the work coordinate system. Once created, the subsequent absolute command refers to the coordinate value in this work coordinate system.

The value x is diameter value in diameter programming and radius value in radius programming.



G50 X 12.87Z37.51* (Diameter programming)

As shown in the above program, the G50 is used to insure that the tool tip coinsides with the program start point and should be programmed at the start of a program.



G50 X60.0Z70.0* (Radius programming)

As shown above a G50 should be at the start of the program to make the standard point on the turret and the program start point coincide.

If an absolute command is performed, the standard point moves along the commanded path. In order to move a tool tip along the programmed path, the distance from tool tip to the standard position must be compensated as tool offset.

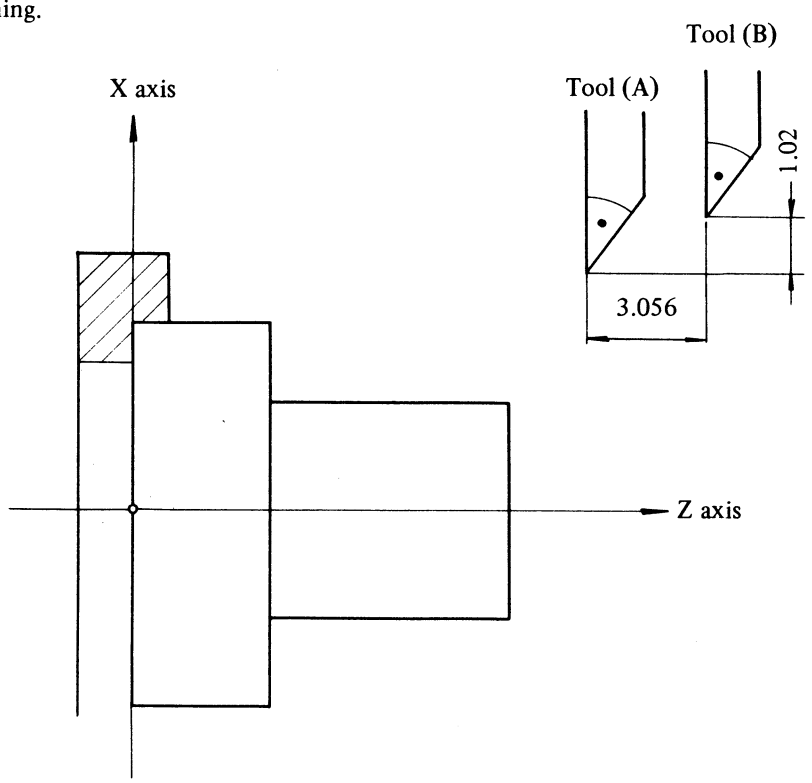
(Note 1) When the coordinate system is set with a G50 command during offset mode, the coordinate system in which coordinate value of a tool excluding the offset value is the specified position is established.

5.7.2 The work coordinate system can be shifted by using the following G50(G92) command

G50U (u) W (w) *

The new, shifted work coordinate system in which the coordinate values of the tool tip 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.

The values, x and u should be diameter value in diameter programming, or radius value in radius programming.



The work coordinate system with same zero point can be set by the following G50 command, when exchanging tool (A) to tool (B) as in the above figure.

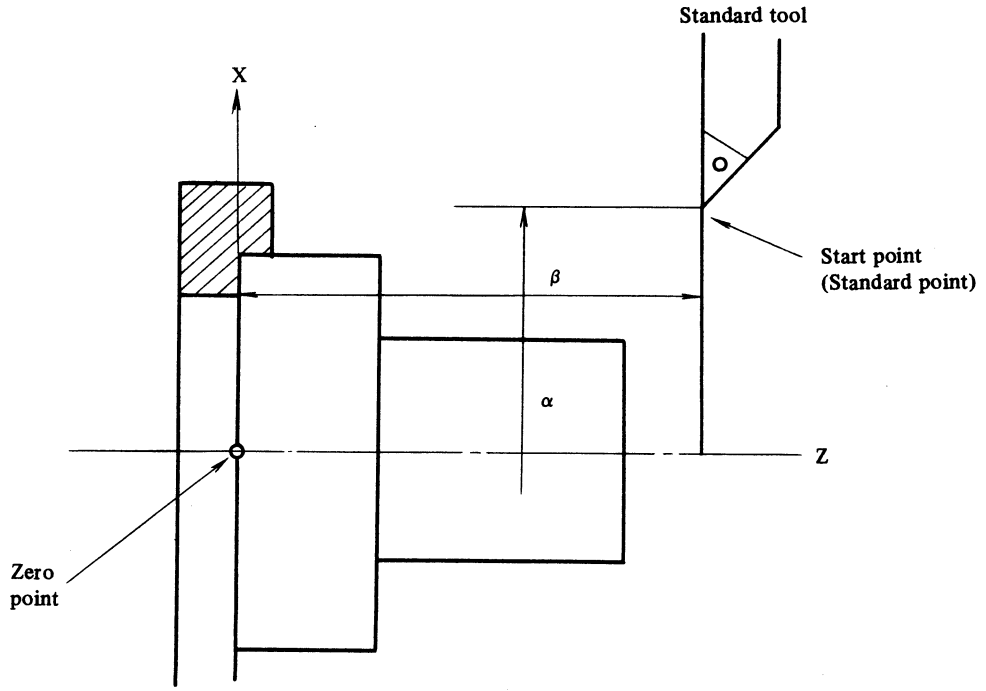
G50U 2.04W 3.056* (diameter programming)

5.7.3 Automatic coordinate system setting

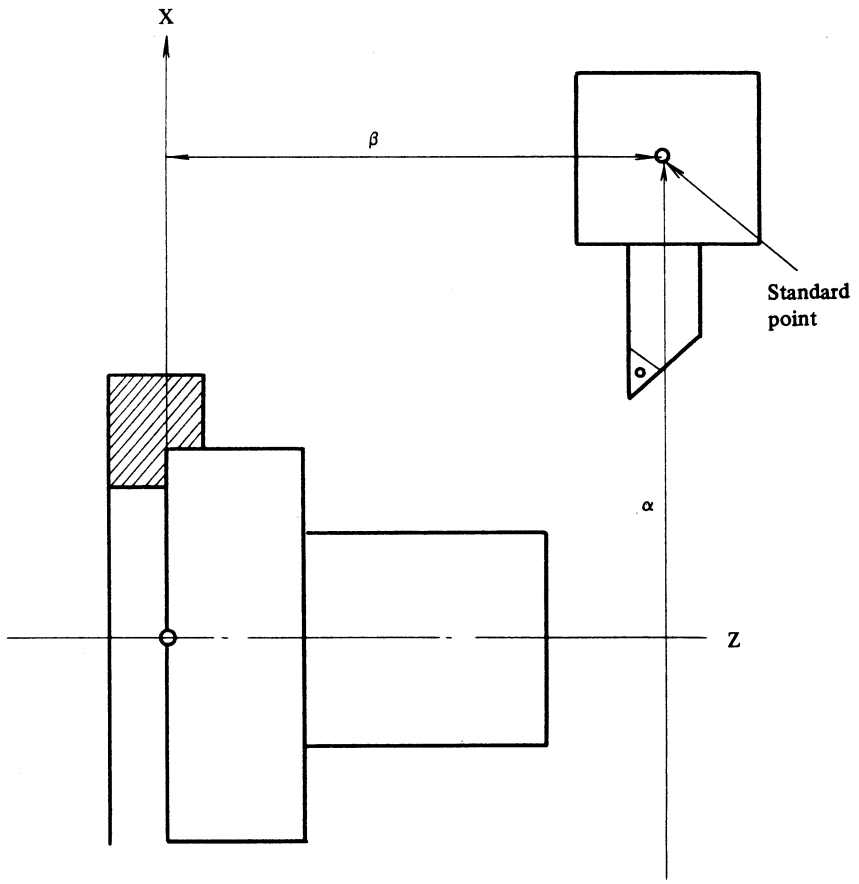
When the manual reference point return is performed, the work coordinate system is automatically set by parameter setting (APRS in parameter number 12). In this case, coordinate values "α" and "β" should be set to the parameters of parameter number 76 and 77 respectively.

If the parameters described in the above are set in advance, the work coordinate system in which the coordinate values of the standard point of turret or the tool tip is "α" and "β" is automatically set in reference point return. It is equivalent that the following G50 command is specified.

G50 X α Z β *



(When the standard point is at the tip of the standard tool)

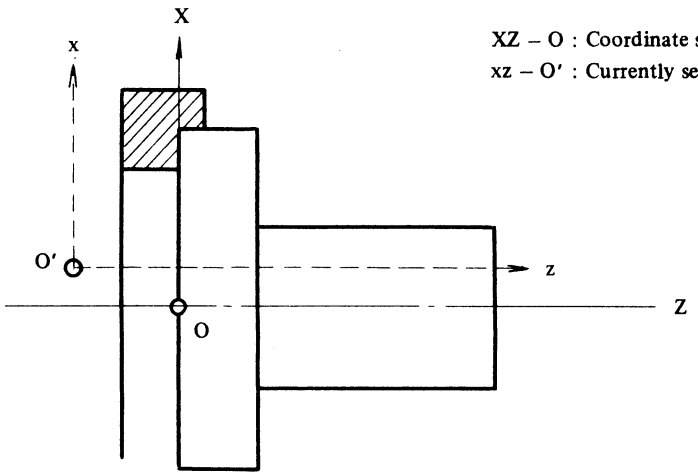


(When the standard point is at the center of the turret.)

5.7.4 Work coordinate system shift

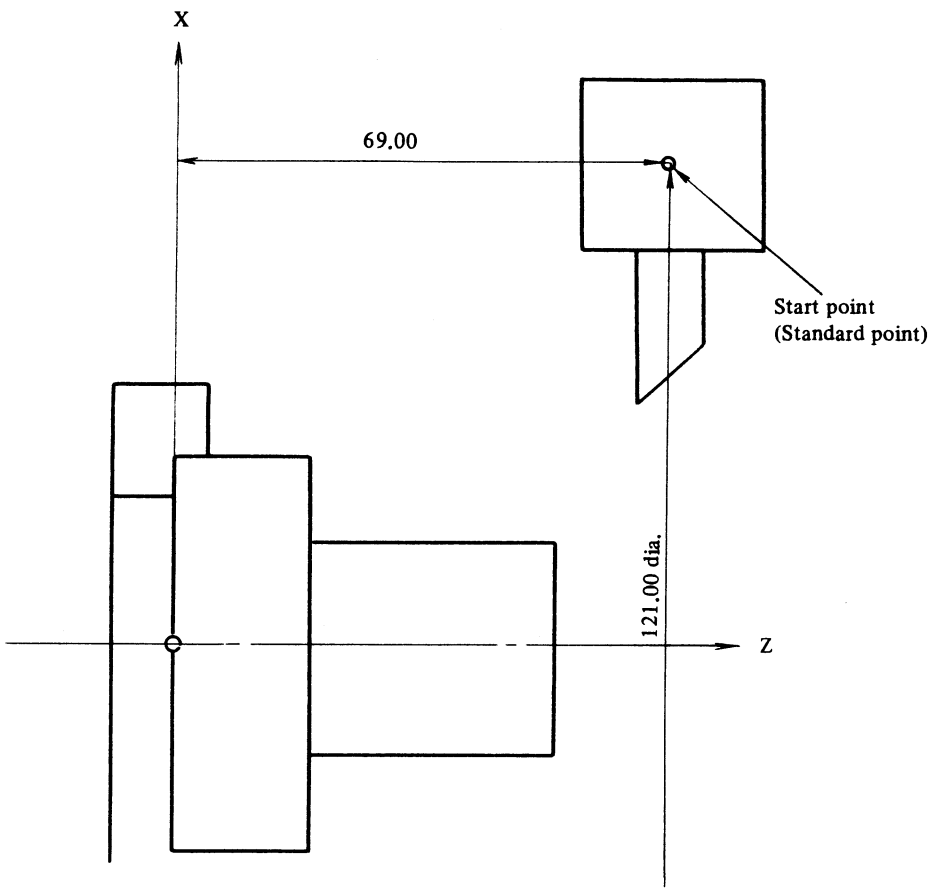
If the work coordinate system set by G50 command or automatic coordinate system setting function is different from the coordinate system in programming, the work coordinate system set in advance can be shifted.

Set the shift amount to the offset memory of offset number 00. Setting procedure of shift amount is same as that in tool offset amount.



XZ – O : Coordinate system in programming
xz – O' : Currently set work coordinate system

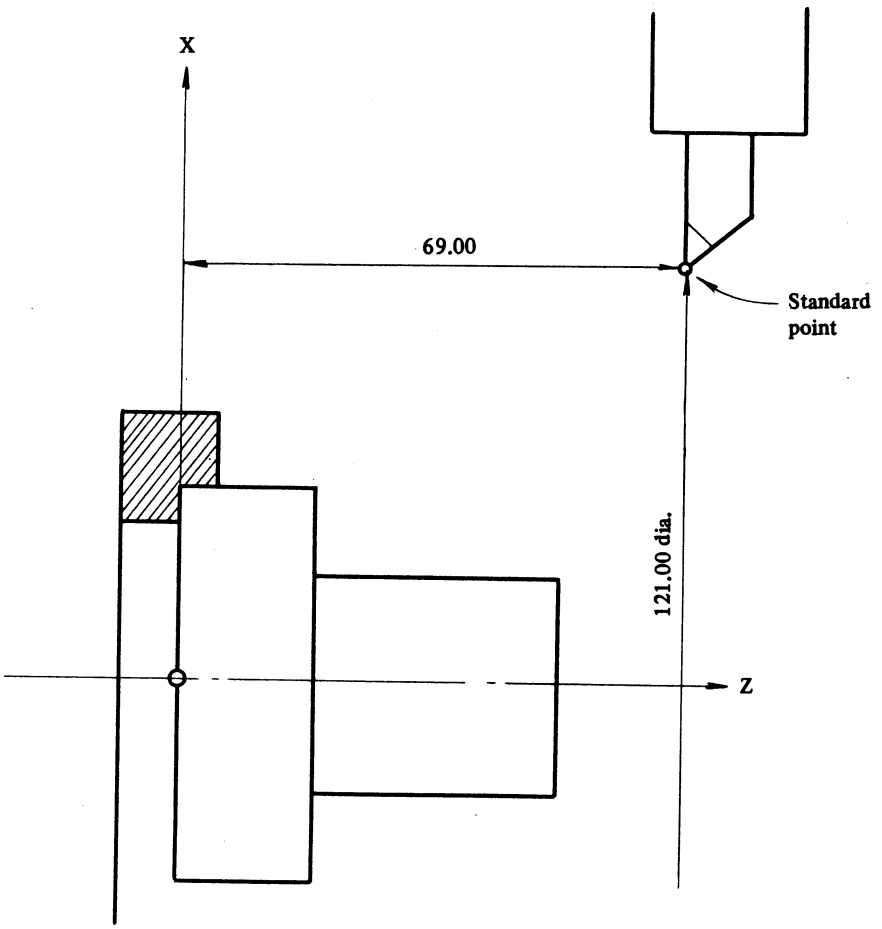
Set the shift amount, O' to O, to the offset memory with number 00.



When the actual standard point is at the position of 121 mm (diameter value) in X axis and 69 mm in Z axis even though the following G50 command is executed.

G50 X 120.0 Z 70.0 *

the desired work coordinate system can be set by shifting the current coordinate system with 1.0 mm in X axis and −1.0 mm in Z axis.



G50 X 120.0 Z 70.0 *
(Diameter programming)

If the standard point is at the tool tip as shown in the above figure, coordinate system can be shifted and the desired work coordinate system can be set. The shift amount is 1.0 mm in X axis and −1.0 mm in Z axis.

At actual setting of shift amount, the method in item 5.7.5 is simple and convenient "Direct measured value input for work coordinate system shift."

(Note 1) The work coordinate system is shifted immediately after setting the shift amount.

(Note 2) After setting the shift amount, if G50 command is programmed, the shift amount become ineffective.

(Example) If the following command is specified, the coordinate values of the current standard point is X = 100.0 and Z = 80.0 being set irrespective of the shift amount previously set.

*G50 X 100.0 Z80.0 **

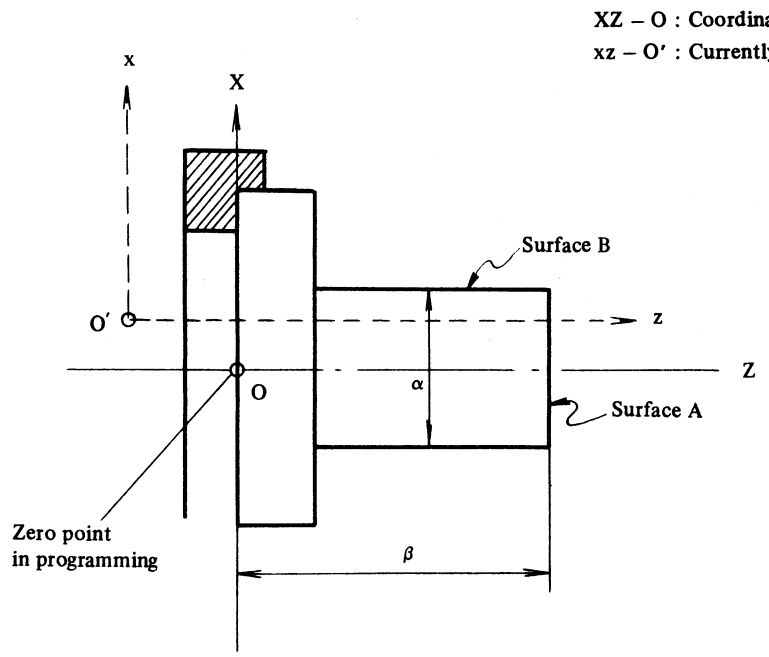
(Note 3) If the automatic coordinate system setting is performed by reference point return after shift amount setting, the coordinate system is shifted instantly.

(Note 4) The work coordinate system shift becomes effective or not by parameter setting (WSFT in parameter number 12).

(Note 5) The shift amount in X axis is programmed in diameter/radius designation depending on that in program.

5.7.5 Direct measured value input for work coordinate system shift

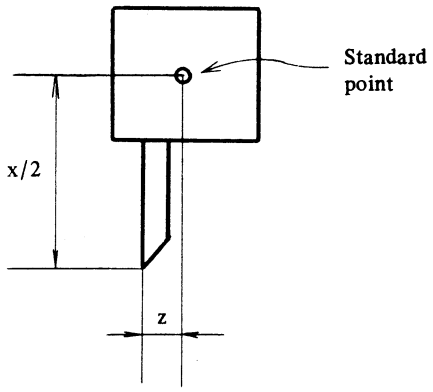
When the work coordinate system set by G50 command or automatic coordinate system setting function is different from the coordinate system in programming, the coordinate system can be shifted by storing the measured distance directly instead of the shift amount as follows:



(1) Cut the workpiece along the surface A using the standard tool in manual operation.
(2) Release the tool along X axis without Z axis movement and stop the spindle rotation.
(3) Measure the distance "β" in the figure above and input it to the offset memory with offset number 100 for Z axis.
(4) Cut the workpiece along the surface B in manual operation.
(5) Release the tool along Z axis without X axis movement and stop the spindle rotation.
(6) Measure the distance "α" and input it to the offset memory with offset number 100 for X axis.

The shift amount, O to O′, is automatically set to the offset memory with offset number 00, and the work coordinate system is shifted immediately.

If the offset amount of standard tool is zero, the work coordinate system in which the coordinate values of the tool tip is X = 0.0 and Z = 0.0 at positioning the tip of the standard tool to the zero point is set.



However, if the tool offset amount is set as shown in the figure above and the tool offset function is effective and the direct measured value input for the work coordinate system shift is performed, the work coordinate system in which the coordinate values of the standard point is X = 0.0 and Z = 0.0 at positioning the standard point to the zero point is set.

(Note 1) The distance "α" should be set in diameter value.

5.8 Inch/Metric Conversion G20, G21 **

Unit system	G code	Least input increment
Inch	G20	0.0001 inch
mm	G21	0.001 mm

These G codes must be commanded by themselves and must be commanded before the work coordinate system setting block at the start of the program.

The followings change depending on these G codes.

(1) Feedrate command by F code
(2) Display of positions
(3) Offset value
(4) Unit of scale on the manual pulse generator
(5) Move distance in incremental feed.

(Note 1) When the power is turned on, G code remains the state before turning power off.

(Note 2) Either inch or metric input can be selected by setting parameter. This setting obeys also G20 or G21 command.

(Note 3) G20 and G21 must not be changed during a program.

(Note 4) When the unit system of a machine and that of programming are different, the maximum switching error is half of the least command increment. The error is not accumulated.

5.9 Switching of Feedrate Command G98, G99 **

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

Refer to item 4.2 for detail

5.10 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.

Refer to item 8.2 for detail.

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

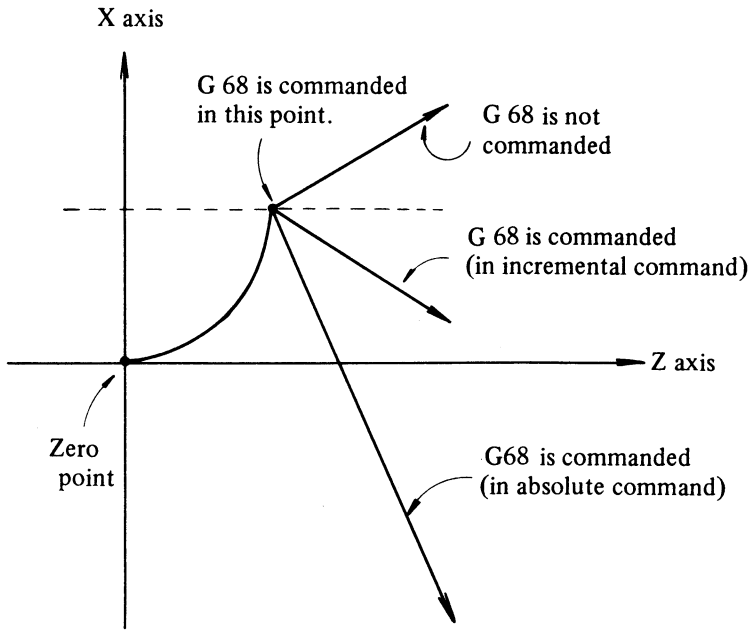
5.11 Program Mirror Image G68, G69 **

Mirror image can be performed on X axis move command by using G code.

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

Commanding G 68 code, the sign of programmed move command on X axis is reversed and the symmetric machining can be performed.

This function is effective against the plane vartical to the X axis from the block after program mirror image is commanded. That is, in incremental command, it is performed against the plane which is vertical to X axis and passes through the start point of the block, while in absolute command, against the plane which passes through the zero point as following figure. This G code should be commanded by itself.



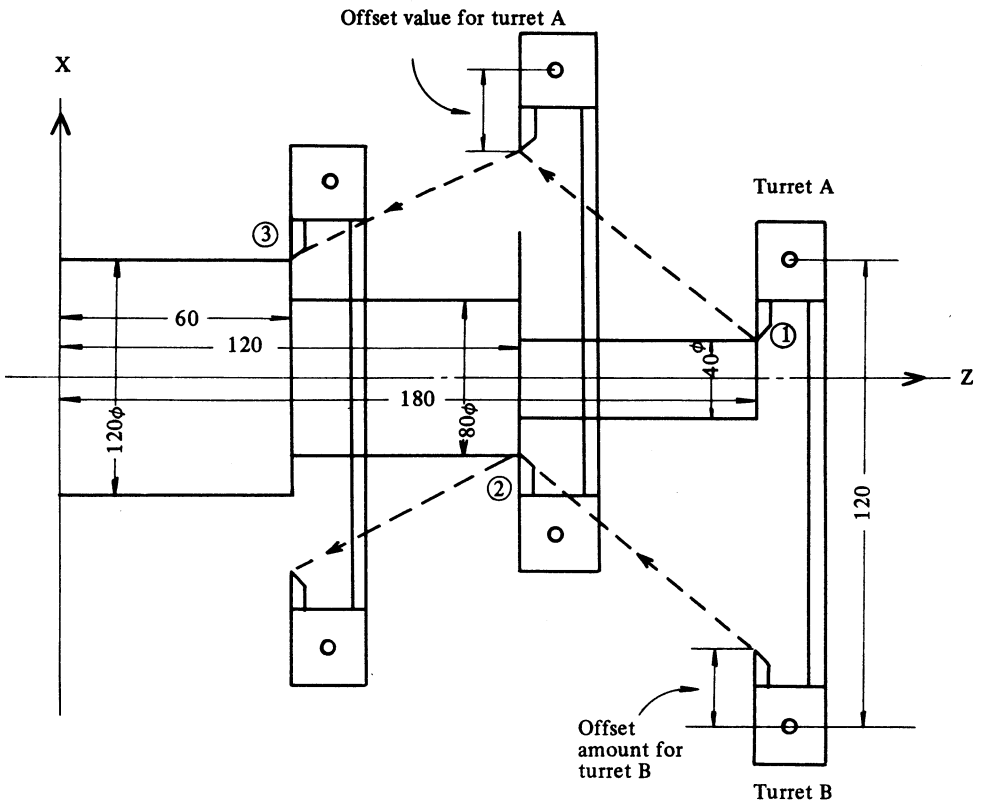
The mirror image function can be cancelled by G69 command. The mirror image function becomes ineffective for the following movement.

(1) Movement in manual operation

(2) Movement from the intermediate point to the reference point in G28 mode (reference point return).

Using this function and work coordinate system shift (Item 5.7.2), NC machining for double turret can be performed by the NC program for one turret.

(Example) Program example for double turrets



X40. Z180. T0101 * ..... Positioning to ① (Turret A).

G50 U-240. * } ..... Coordinate system shift by the distance between
G68 * } Turret A and B, and mirror image ON.

X80. Z120. T0202 * ..... Positioning to ② (Turret B)

G50 U-240. * } ..... Coordinate system shift by the distance between
G69 * } Turret A and B, and mirror image cancel.

({ G69* / G50 U-240.* } is available.)

X120. Z60. T0101 * ..... Positioning to ③ (Turret A).

6. COMPENSATION FUNCTION

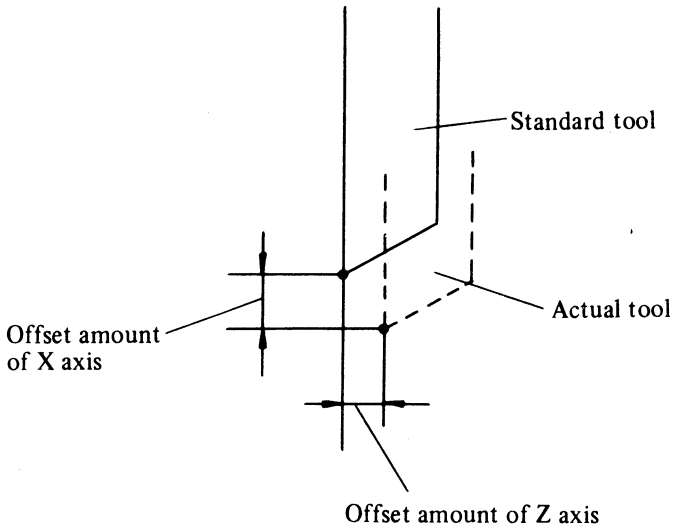
When the tool different from the imagined tool is used in programming, the compensation function is used. This function is tool offset.

6.1 Tool Offset

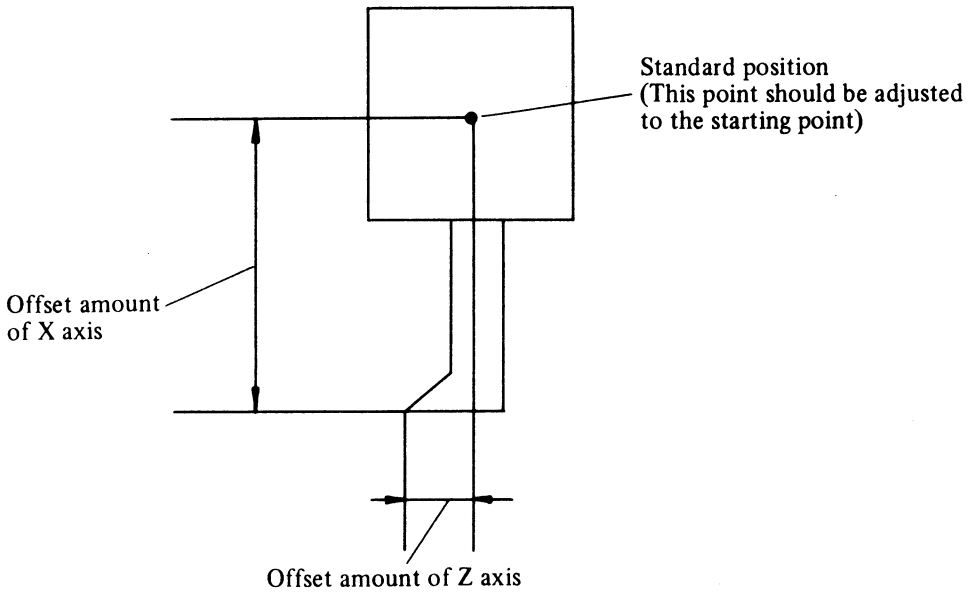
In this system, tool offsets can be controlled only by T codes. It is not controlled by G codes.

6.1.1 Basic tool offset

Tool offset is used when the tool actually used differs from the standard tool in programming.



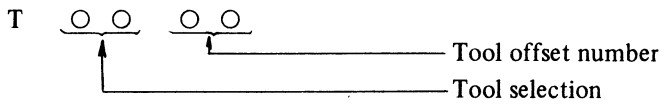
Programming is done so that the tip of the standard tool moves over the desired path, and the tool is set to match the standard tool starting point in the program. However, the actual tool used seldom matches the standard tool exactly. Rather than attempting to set the tool tip position, it is more convenient to set the standard point of the machine, such as the center of the turret. In this case, the distance between the standard position and the tool tip position is used as the offset amount.



The standard position can be determined in two ways: the point that is the starting position on the reference point can be taken as the standard position, or a convenient reference point can be made the starting point of the program.

6.1.2 T code for tool position offset

T codes have the following meaning.



(Note 1) According to machine, tool selection is set by 1-digit.

6.1.3 Tool selection

Tool selection is made by specifying the T code corresponding to the tool number.

Refer to the manual issued by the machine tool builder for the relationship between the tool selection number and the tool.

6.1.4 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 00 means that the offset amount is 0 and the offset is cancelled.

The offset distance must be set into the offset memory by the MDI & DPL unit or G10 command in correspondence to the offset number.

Two offset amount can be set to the corresponding offset number, one of them is that for X axis and the other for Z axis.

Offset number	OFS	
	X axis offset amount	Z axis offset amount
01	0.040	0.020
02	0.060	0.030
03	0	0
04	.	.
05	.	.
.	.	.
.	.	.

Tool offset is effective when the T code is specified and its offset number is not 00.

If it is 00, the tool offset function is cancelled.

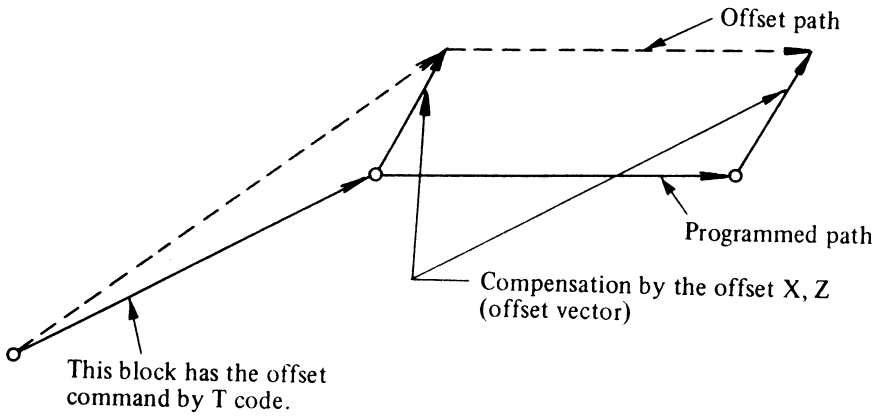
Values that can be set as the offset distance are as follows:

mm input	0 ~ ±999.999 mm
inch input	0 ~ ±99.9999 inch

Parameter (ORC) setting can switch the axis tool offset amount of X axis to either diameter designation or radius designation.

6.1.5 Offset

Offset X, Z is only offset for the programmed path. The offset distance corresponding to the number specified by the T code is added to or subtracted from the end position of each programmed block.



6.1.6 Offset vector

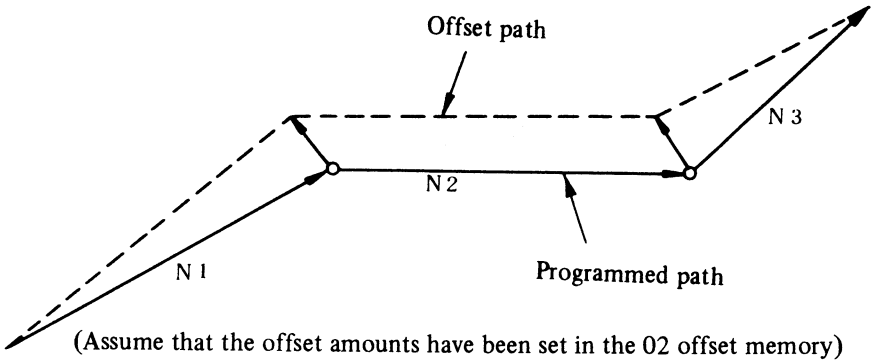
In the diagram above, the vector with offset X, Z is called the offset vector. The compensation is the same as the offset vector.

6.1.7 Offset cancel

Offset is cancelled when the T code offset number 00 is selected. At the end of the cancelled block,

the offset vector becomes zero.

N1 U 50.0 W 100.0 T0202*
N2 W 100.0*
N3 U 50.0 W 50.0 T0200*



(Assume that the offset amounts have been set in the 02 offset memory)

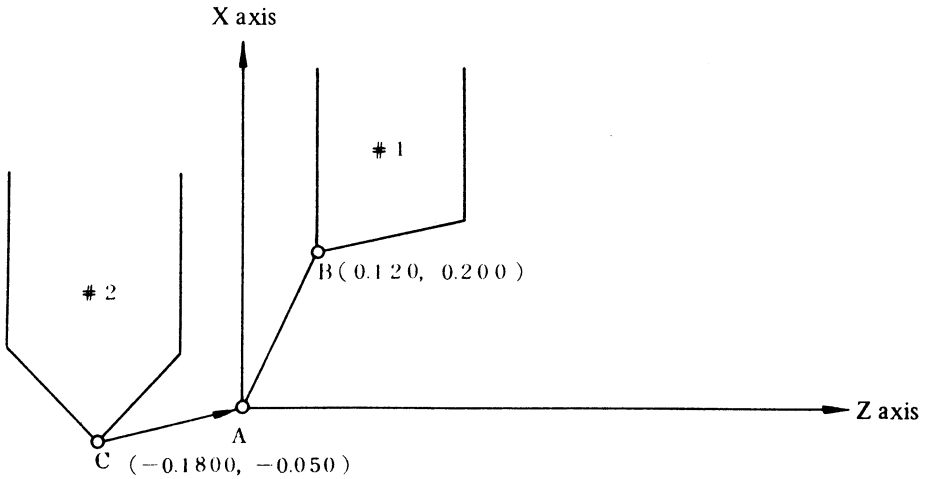
When the power is turned on, the reset key on the MDI unit is pushed or the rest signal is input to the NC from the machine, the offset is cancelled and the offset number becomes 00.

Parameter setting (TOC) can be set so that the offset will not be cancelled by reset key or reset signal input.

(Note 1) When reference point return is performed by manual operation or G28, the offset vector of the axis reaching the reference point is cancelled.

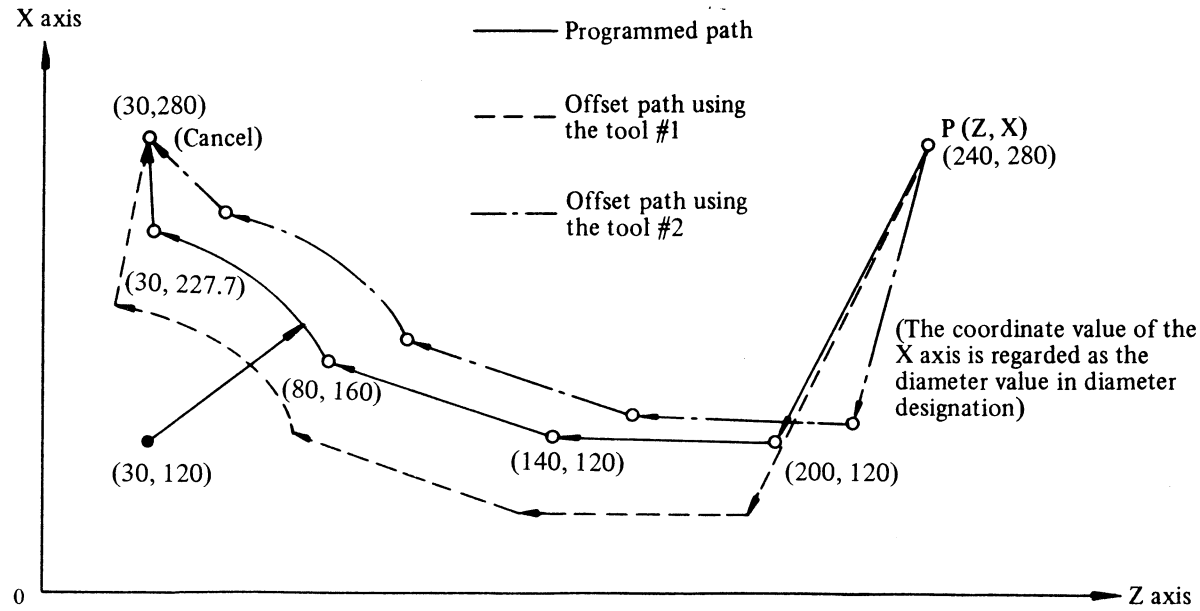
6.1.8 Program example

Tool tip coordinate value (Z, X)	Tool number
Tool #1 ..... B (0.120, 0.200)	01
Tool #2 ..... C (–0.180, –0.050)	02



A: Programmed tool nose
B: Actual tool nose (#1)
C: Actual tool nose (#2)

	Tool number	
	01	02
X	−0.200	+0.050
Z	−0.120	+0.180



(Program example 1)

```
G50 X 280.0 Z 240.0*
G00 X 120.0 Z 200.0 T 0101*
G01 Z 140.0 F 30*
    X 160.0 Z 80.0*
G03 X 227.7 Z 30.0 R53.81*
G00 X 280.0 T0100*
```

The path of the tool tip of tool #1 coincides with the programmed path by this program.

(Program example 2)

The path of the tool tip of tool #2 will coincide with the programmed path by the making following changes in program example 1.

T0101 → T0202
T0100 → T0200

6.1.9 Only T code

When only a T code is commanded in a block, the offset movement is performed without a move command. The movement is performed at rapid traverse in G00 mode, while at cutting feed in other modes.

When T code with offset number 00 is commanded by itself, movement is performed to cancel the offset.

(Note 1) *G50 X (x) Z (z) T__**
Tool movement is not performed.
The coordinate system in which the coordinate value of tool position is (x, z) is set. The tool position is the result of the subtraction of the offset amount corresponding to offset number specified by T code.

(Note 2) *G40T__*, G02 ..... T__* and G10 ..... T__**
Only the tool change is performed and the tool offset is not performed.

(Note 3) *When the offset amount corresponding with offset number being used in the automatic operation is changed by MDI operation, the new offset amount is not available until the T code of the offset number is again commanded.*

6.2 Program Input of Offset Amount G10 **

The offset amount in tool offset can be input by program, G10 command.

G10 P____X ____Z ____* (absolute command)

G10 P ____U ____W ____* (incremental command)

P : Offset number
X : Offset amount in X axis (in absolute)
Z : Offset amount in Z axis (in absolute)
U : Offset amount in X axis (in incremental)
W : Offset amount in Z axis (in incremental)

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

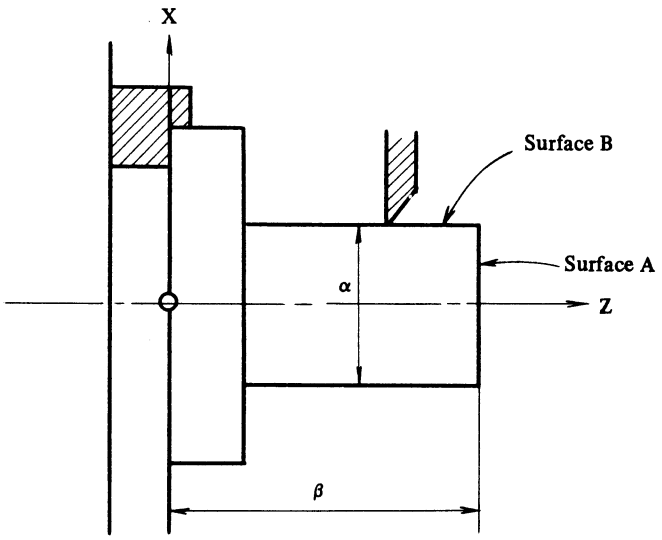
In incremental command, the value specified by addresses U and W are added to the stored offset amount corresponding to the offset number.

(Note 1) *It is not necessary that addresses in a block are absolute address only or incremental address only.*

(Note 2) *By using this command in machining program, a tool can be moved little by little in actual machining.*

6.3 Direct Measured Value Input for Tool Offset

The following convenient method is available in setting the offset amount that the difference between the standard point, such as the tip of the standard tool or the turret center, and the tip of the tool actually used.



(1) Cut the workpiece along the surface A in manual mode.
(2) Release the tool along X axis without Z axis movement and stop the spindle rotation.
(3) Measure the distance "β" between the surface A and the zero point in the work coordinate system, and set it at the offset number which is added one hundred to the offset number desired to set the offset amount.
(4) Cut the workpiece along the surface B in manual mode.
(5) Release the tool along Z axis without X axis movement and stop the spindle rotation.
(6) Measure the distance "α" and set it to the offset number explained in step (3).

For example, set "α" and "β" in offset number 103 so that the offset amount is set to the offset number 03.

If the coordinate value of the tool at the surface B is 105.0, the measured distance is 104.0 and 104.0 is set to the offset number 103, the offset amount 1.0 is automatically set to the offset number 03.

(Note 1) The direct measured value input for tool offset becomes effective or not depending on the parameter setting (DOFSI of parameter number 12).

(Note 2) The distance "α" should be set in diameter value.

7. MACHINING CYCLE FUNCTION **

For repetitive machining peculiar to turning, such as metal removing in rough cutting, a series of paths that is specified usually in a range of three to dozens of blocks, can be specified in one block. In addition, for repetition of the operation, only the values to be exchanged need to be specified, and the program using this cycle is very simple and useful.

Note that the drawings shown in the examples below are illustrated in diameter programming. In radius programming, change parameters U/2 or X/2 to U or X respectively.

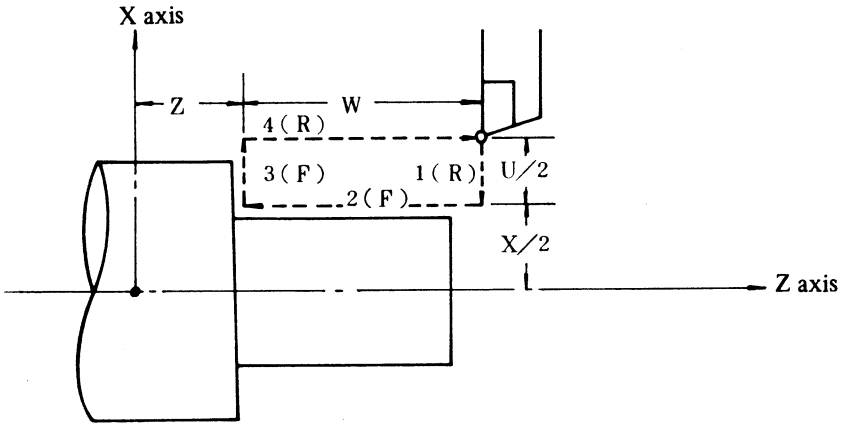
7.1 Canned Cycle

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

7.1.1 Cutting cycle A, G90

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

G90X(U) —— Z(W) —— F —— *

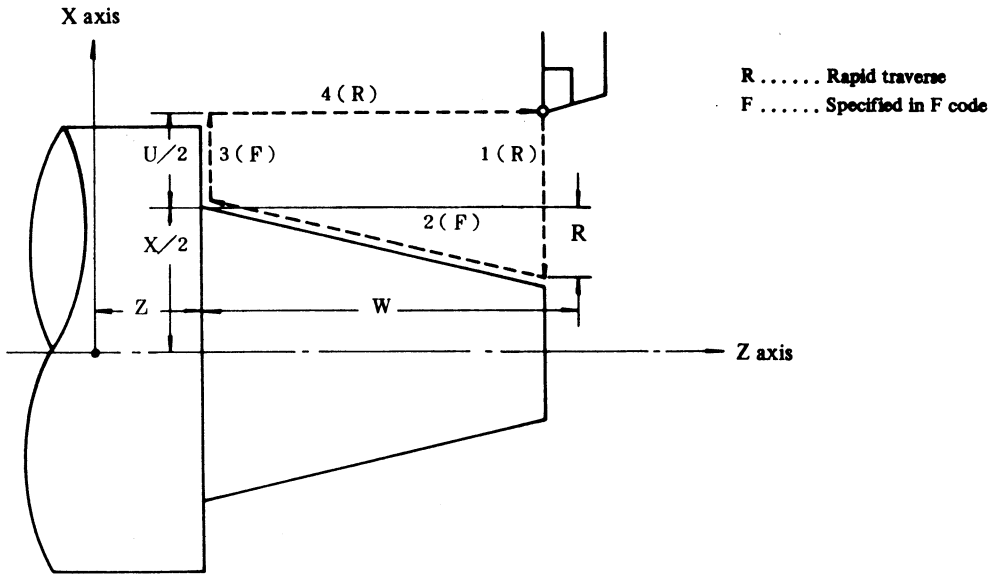


R ..... Rapid traverse
F..... Specified in F code

In incremental programming, the sign of the numbers following address U and W depends on the direction of pathes 1 and 2. In the above example, the signs of U and W are negative. In single block mode, operation 1, 2, 3 and 4 are performed by pressing cycle start once.

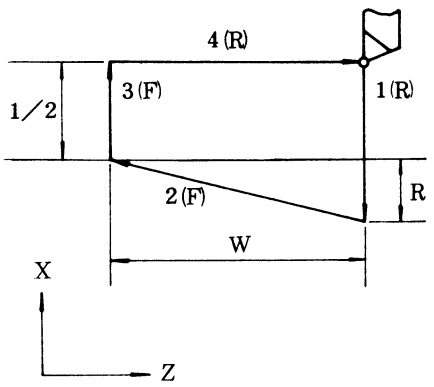
(2) The taper cutting cycle is programmed by the following command.

G90X(U) —— Z(W) —— R —— F —— *

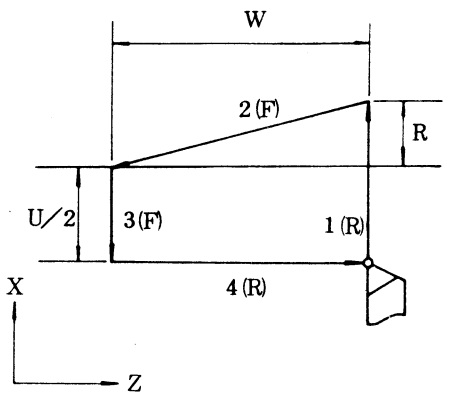


In incremental programming, the relationship between the signs of the numbers following addresses U, W and R, and the tool paths are as follows:

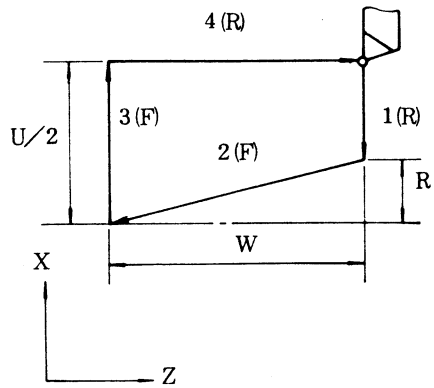
(i) U<0, W<0, R<0



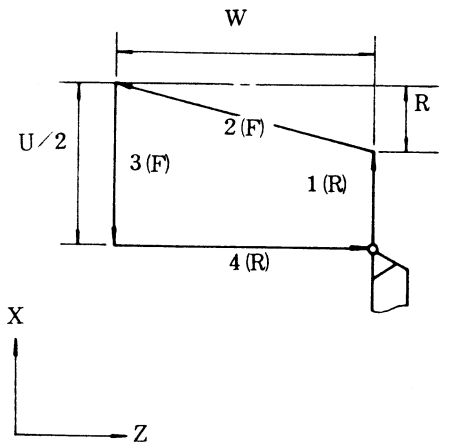
(ii) U>0, W<0, R>0



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



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

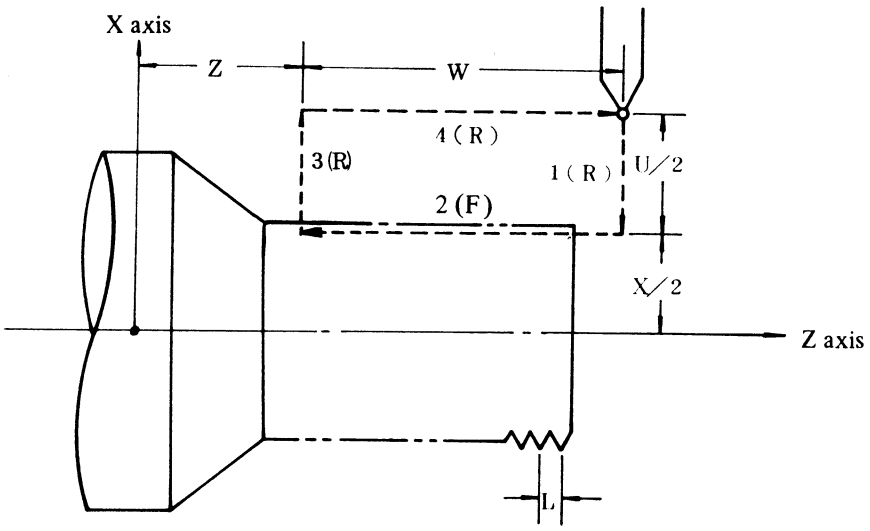


7.1.2 Thread cutting cycle, G92

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

G92X(U) —— Z(W) —— F ——*

Lead (L) is specified.



R ..... Rapid traverse
F ..... Specified in F code

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

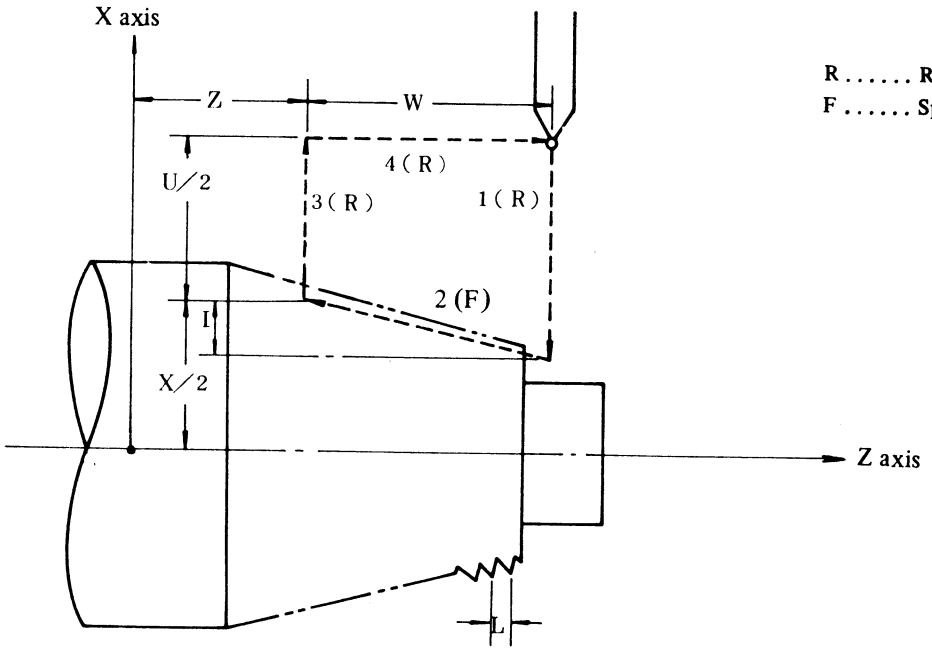
In the single block mode, operations 1, 2, 3 and 4 are performed by pressing cycle start once.

(Note 1) Notes on this thread cutting are the same as in thread cutting in G32.
However, a stop by feed hold is as follows: Stop after completion of path 3 of thread cutting cycle.

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

G92X(U) —— Z(W) —— R —— F —— *

Lead (L) is specified.

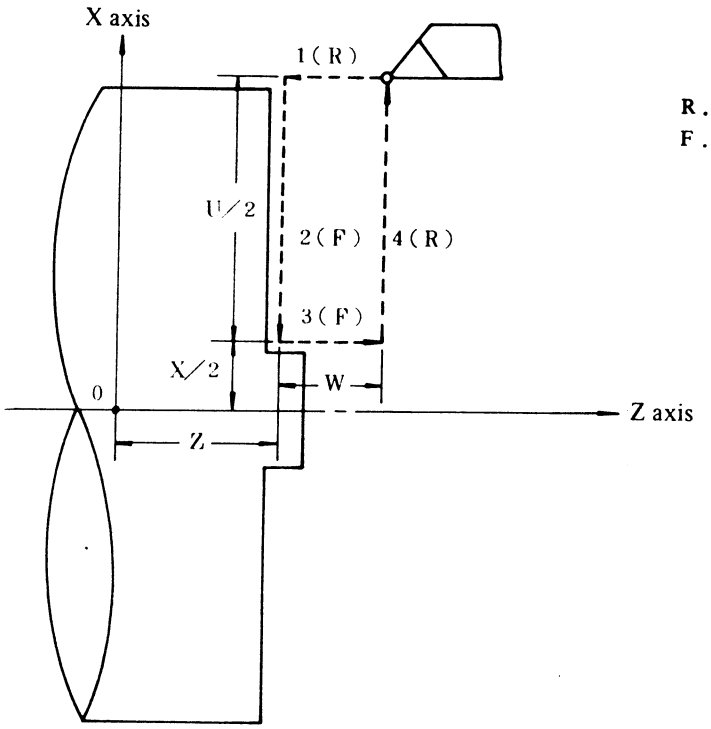


R ..... Rapid traverse
F ..... Specified in F code

7.1.3 Cutting cycle B, G94

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

G94X(U) — Z(W) — F — *



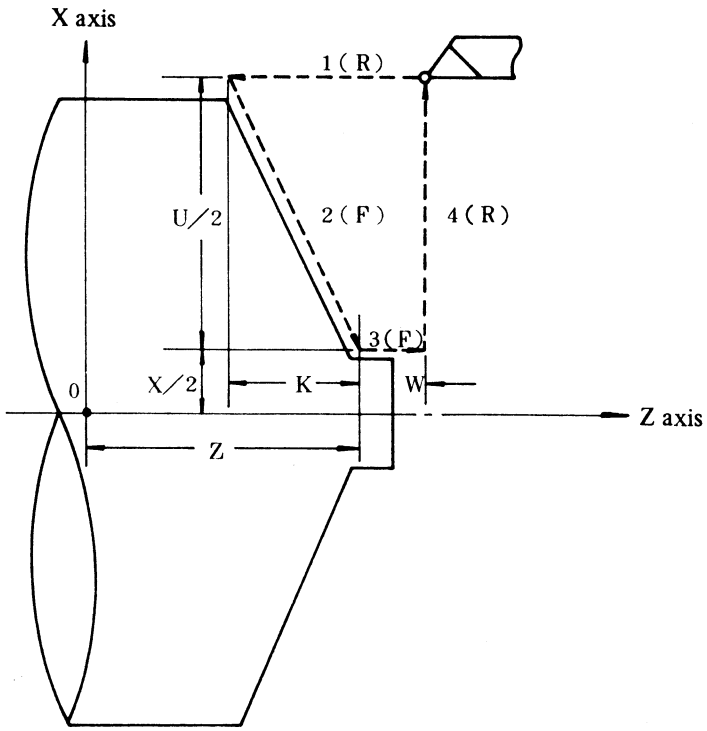
R ..... Rapid traverse
F ..... Specified in F code

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

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

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

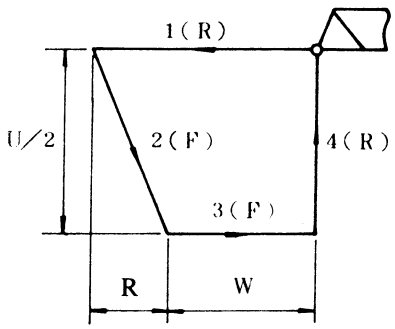
G94X(U) —— Z(W) —— R —— F —— *



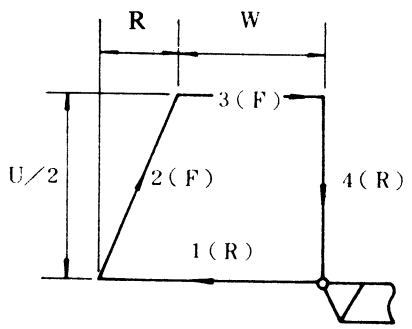
R ..... Rapid traverse
F ..... Specified in F code

In incremental programming, the relationship between the sign of numbers following addresses U, W and R, and the tool path is as follows.

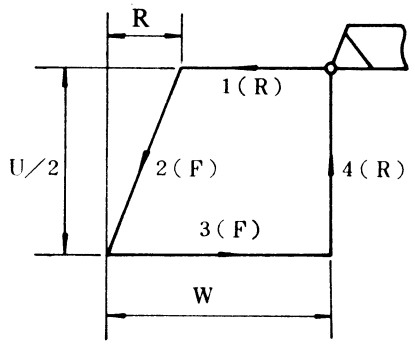
(i) $U<0, W<0, R<0$



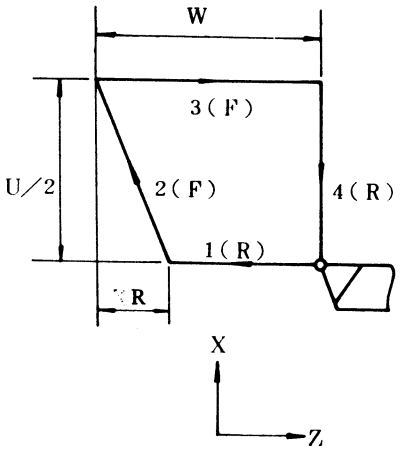
(ii) $U>0, W<0, R<0$



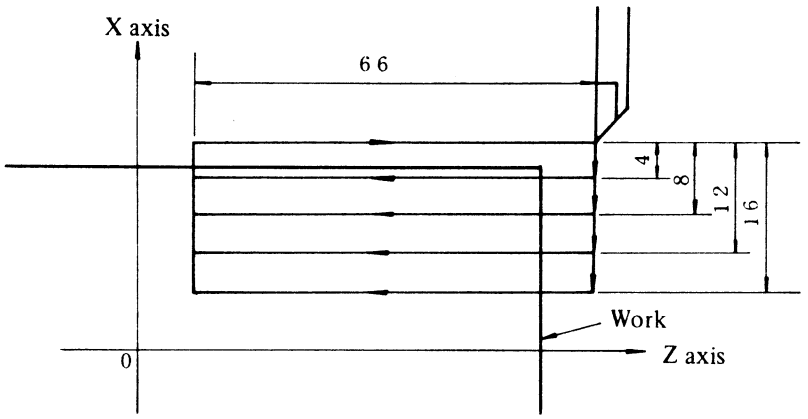
(iii) U < 0, W < 0, R > 0 at |R| ≦ |W|



(iv) U > 0, W < 0, R > 0 at |R| ≦ |W|



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



[Unit: mm]

The cycle in the above figure is executed by the following program (Diameter programming)

N030 G90	*U-8000 W-66000F400**
N031	*U-16000**
N032	*U-24000**
N033	*U-32000**

(Note 2) The following three applications can be performed.

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

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

(3) If the M, S or T function is commanded during the canned cycle mode, both the canned cycle and M, S or T function can be performed simultaneously. If this is inconvenient, cancel the canned

cycle once as in the program examples below (specify G00 or G01) and execute the M, S or T command. After the execution of M, S or T terminates, command the canned cycle again.

*Example: N003T0101**

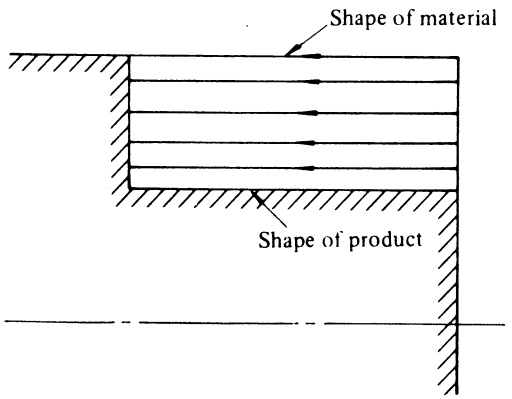
⋮

*N010 G90X 20000 Z10000 F200**
*N011 G00T0202**
*N012 G90X 20500 Z10000**

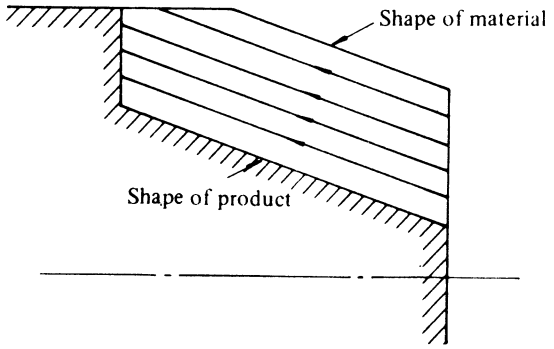
7.1.4 Usage of canned cycle

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

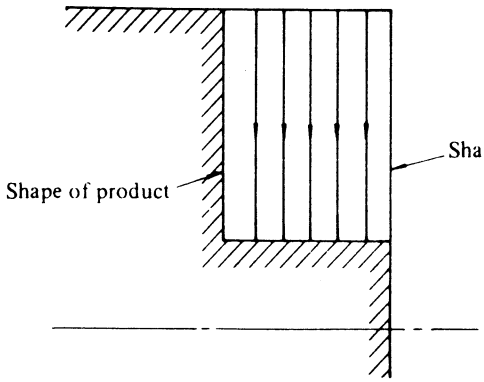
(1) Straight cutting cycle



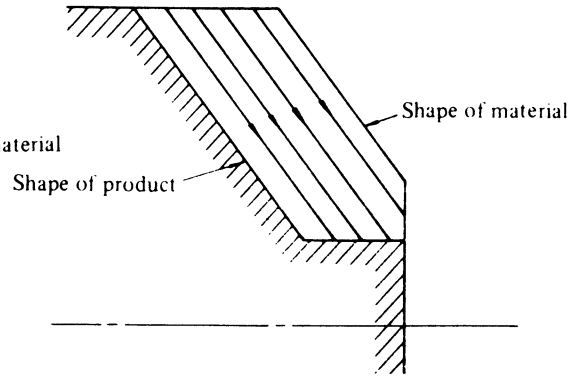
(2) Taper cutting cycle



(3) Face cutting cycle



(4) Face taper cutting cycle

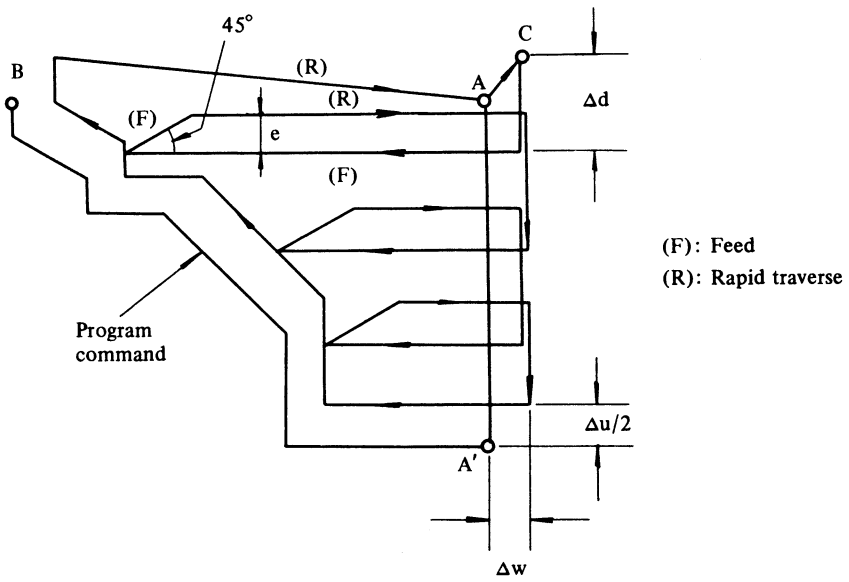


7.2 Multiple Repetitive Cycle G70∿G76

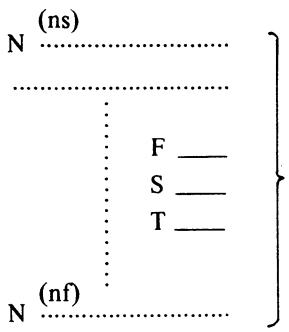
This option provides canned cycles to make NC programming easy. For instance, the data of the finished work shape describes the tool path for rough machining. And also, a canned cycle for the thread cutting is available.

7.2.1 Stock removal in turning (G71)

If a finished shape of A to A′ to B is given by a program as in the figure below, the specified area is removed by Δd, with finishing allowance Δu/2 and Δw left.



G71 U(Δd) R(e) *
G71 P(ns) Q(nf) U(Δu) W(Δw) F(f) S(s) T(t) *



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

Δd : Depth of cut (radius designation)
Designate without sign. The cutting direction depents on the direction AA′.
This designation is modal and is not changed until the other value is designated. Also this value can be specified by the parameter (No. 82), and the parameter is changed by the program command.

e : Escaping amount
This designation is modal and is not changed until the other value is designated. Also this value can be specified by the parameter (No. 83), and the parameter is changed by the program command.

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

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

Δu : Distance and direction of relief in X direction (diameter/radius designation).

Δw : Distance and direction of relief in Z direction.

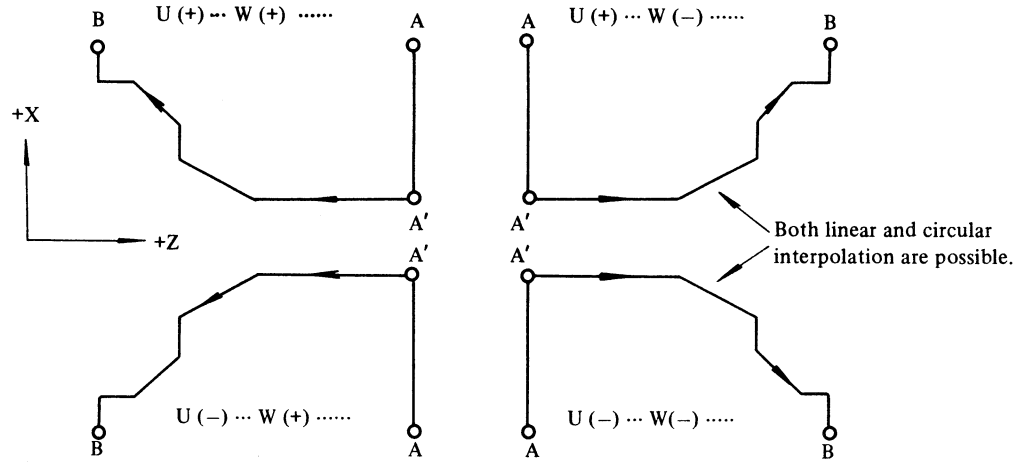
f, s, t : Any F, S or T function contained in blocks of ns to nf in the cycle is ignored, and the F, S or T function in this G71 block is effective.

(Note 1) While both Δd and Δu are specified by address U, the meanings of them are determined by the presence of address P and Q.

(Note 2) The cycle machining is performed by G71 command with P and Q specification.

F, S and T functions which are specified in the move command between points A and B are ineffective and those specified in G71 block or the previous block are effective. When an option of constant surface speed control is selected, G96 or G97 command specified in the move command between points A and B are ineffective, and that specified in G71 block or the previous block is effective.

The following four cutting patterns are considered. All of these cutting cycles are made parallel to Z axis and the sign of Δu and Δw are as follows:

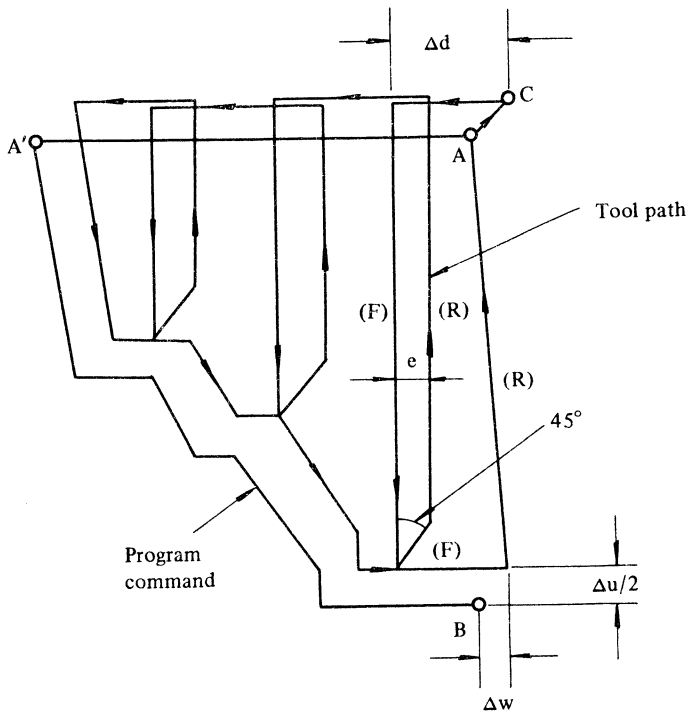


The tool path between A and A′ is specified in the block with sequence number "ns" including G00 or G01, and in this block, a move command in the Z axis cannot be specified. The tool path between A′ and B must be steadily increasing or decreasing pattern in both X and Z axis. When the tool path between A and A′ is programmed by G00/G01, cutting along AA′ is performed in G00/G01 mode respectively.

(Note 3) The subprogram cannot be called from the block between sequence number "ns" and "nf".

7.2.2 Stock removal in facing (G72)

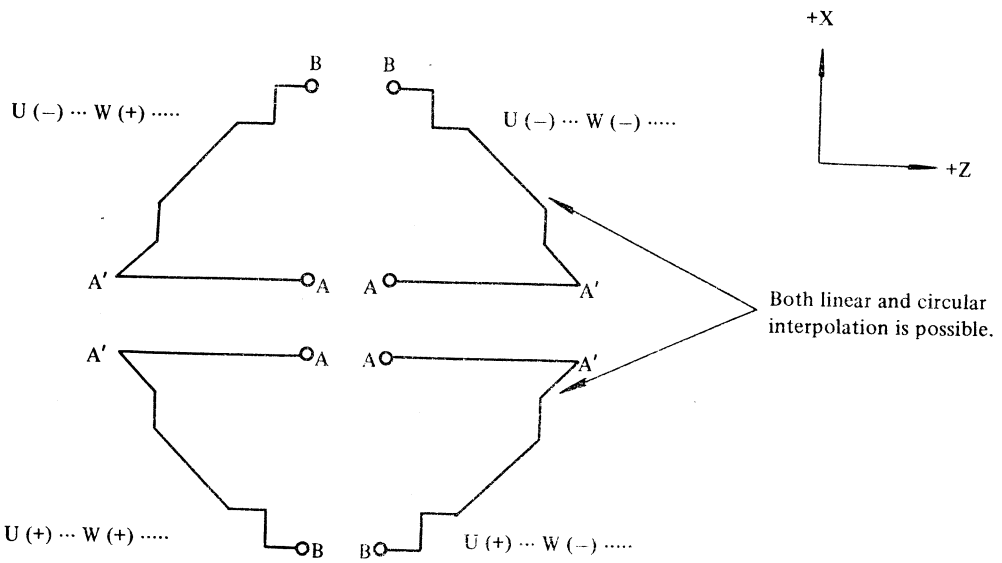
As shown in the figure below, this cycle is the same as G71 except that cutting is made by a operation parallel to X axis.



G72 W(Δd) R(e) *
G72 P(ns) Q(nf) U(Δu) W(Δw) F(f) S(s) T(t) *

The meanings of Δd, e, ns, nf, Δu, Δw, f, s, and t are the same as those in G71.

The following four cutting patterns are considered. All of these cutting cycles are made paralle to X axis and the sign of Δu and Δw are as follows.

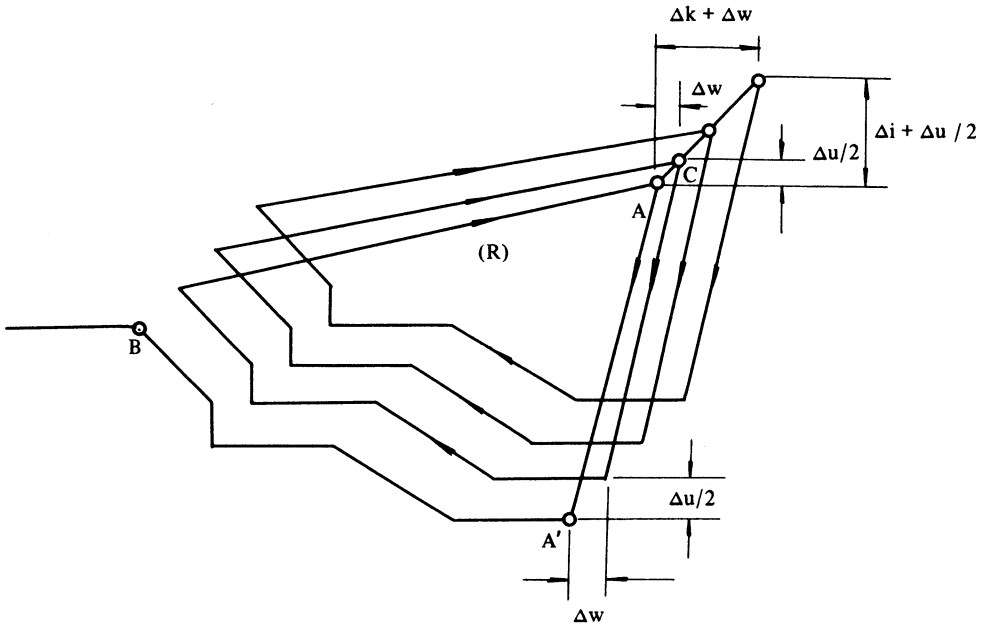


The tool path between A and A′ is specified in the block with sequence number "ns" including G00 or G01, and in this block, a move command in the X axis cannot be specified. The tool path between A′ and B must be steadily increasing and decreasing pattern in both X and Z axis.

Whether the cutting along AA′ is G00 or G01 mode is determined by the command between A and A′, as described in item 7.2.1.

7.2.3 Pattern repeating (G73)

This function permits cutting a fixed pattern repeatedly, with a pattern being displaced bit by bit. By this cutting cycle, it is possible to efficiently cut work whose rough shape has already been made by a rough machining, forging or casting method, etc.



The pattern commanded in the program should be as follows.

A → A′ → B

G73 U(Δi) W(Δk) R(d) *
G73 P(ns) Q(nf) U(Δu) W(Δw) F(f) S(s) T(t) *

N (ns) .....
.....
N (nf) .....

The commands between A → A′ → B are specified in the blocks between sequence number "ns" and "nf".

Δi : Distance and direction of relief in the X axis direction. (radius designation).
This designation is modal and is not changed until the other value is designated. Also this value can be specified by the parameter (No. 84), and the parameter is changed by the program command.

Δk : Distance and direction of relief in the Z axis direction.
This designation is modal and is not changed until the other value is designated. Also this value can be specified by the parameter (No. 85), and the parameter is changed by the program command.

d : The number of division
This value is the same as the repetitive count for rough cutting. This designation is modal and is not changed until the other value is designated. Also, this value can be specified by the parameter (No. 86), and the parameter is changed by the program command.

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

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

Δu : Distance and direction of relief in X direction (diameter/radius designation)

Δw : Distance and direction of relief in Z direction.

f, s, t : Any F, S and T functions contained in the blocks between sequence number "ns" and "nf" are ignored, and the F, S and T functions in this G73 block are effective.

(Note 1) *While the values Δi and Δk, or Δu and Δw are specified by address U and W respectively, the meanings of them are determined by the presence of addresses P and Q in G73 block.*

(Note 2) *The cycle machining is performed by G73 command with P and Q specification.*

The four cutting patterns are considered. Take care of the sign of Δu, Δw, Δk, and Δi.
When the machining cycle is terminated, the tool returns to point A.

7.2.4 Finishing cycle

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

G70 P (ns) Q (nf) *

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

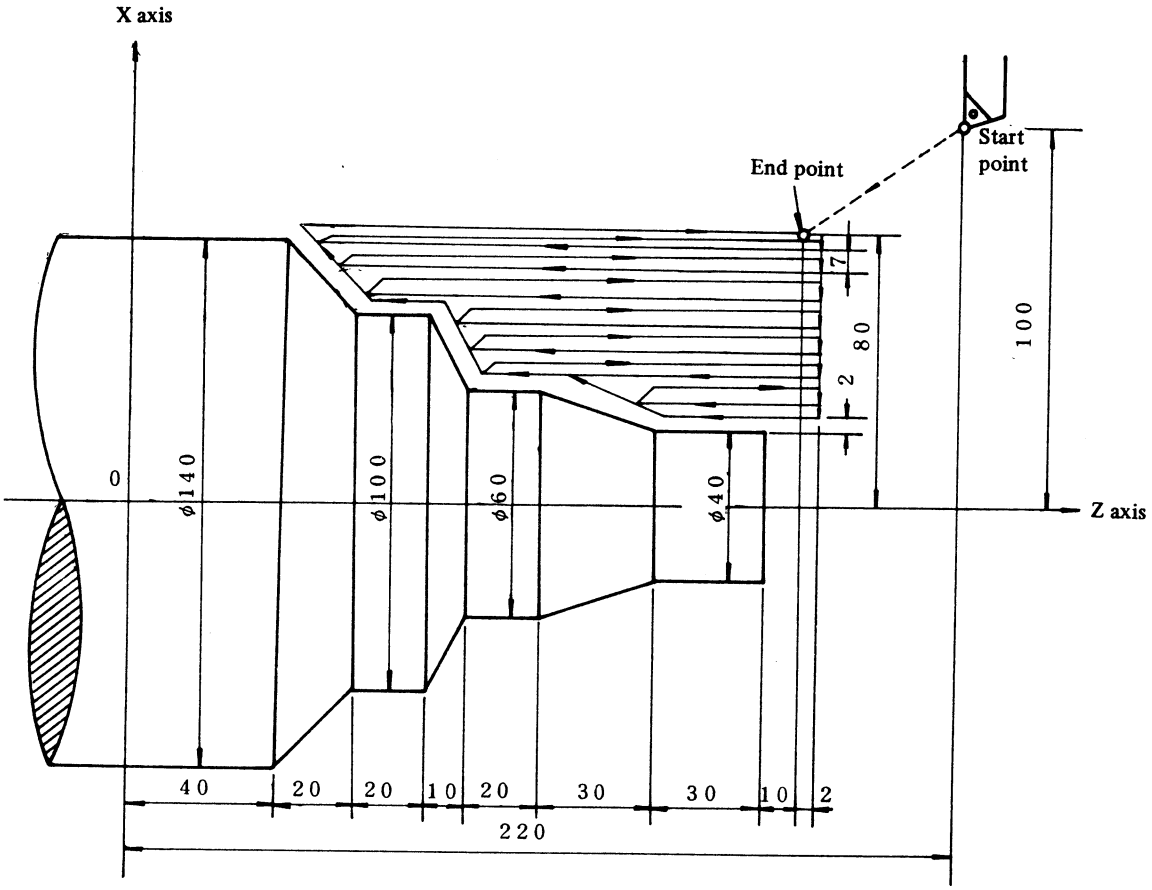
nf : Sequence number of the last block for the program of finishing shape.

(Note 1) *F, S and T functions specified in the block G71, G72, or G73 are not effective but those which are specified between sequence numbers "ns" and "nf" are effective in G70.*

(Note 2) *When the cycle machining by G70 is terminated, the tool is returned to the start point and the next block is read.*

(Note 3) *In blocks between "ns" and "nf" of G70 through G73, the subprogram cannot be called.*

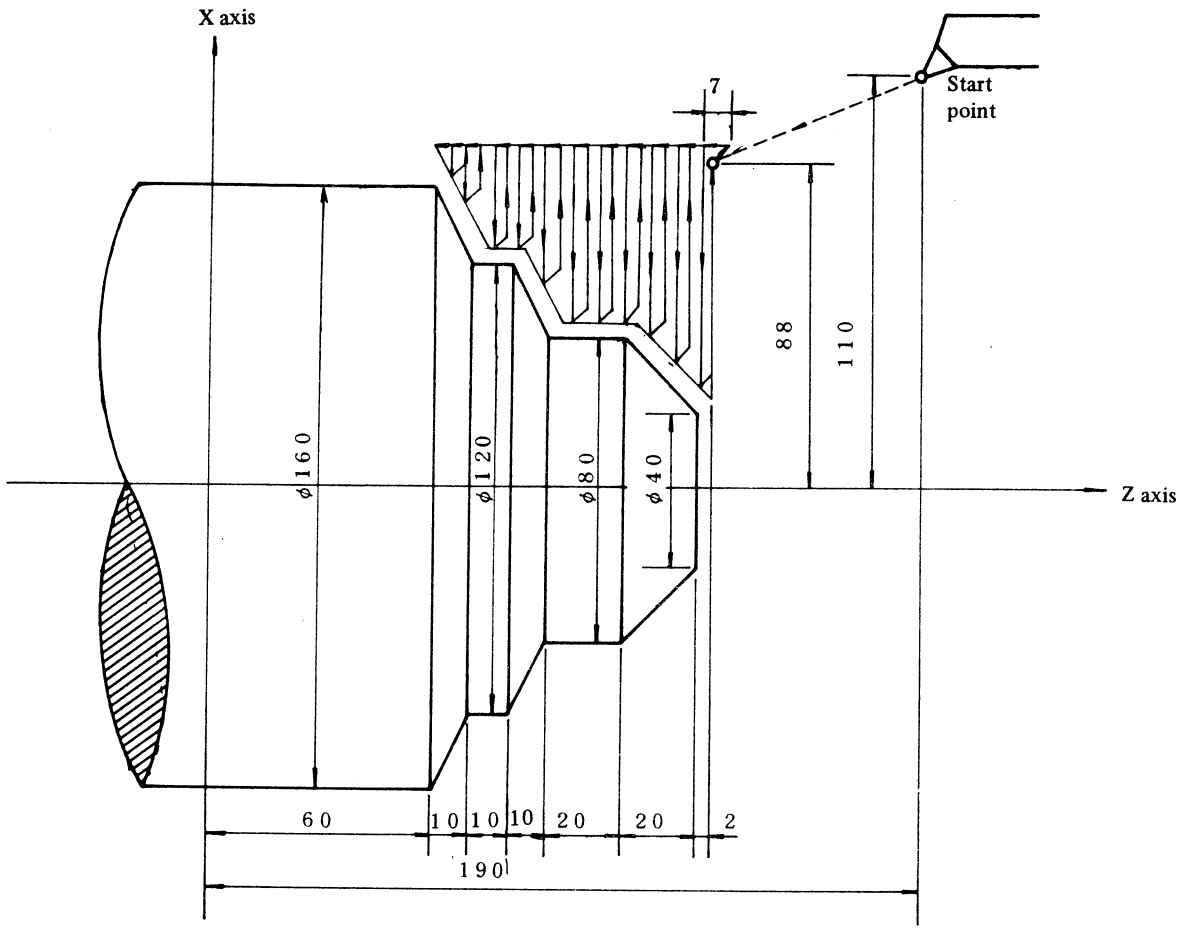
(Ex. 7.2.1) Example of programming by multiple repetitive cycle (G70, G71)



(Diameter designation, metric input)

```
N010G50X200.0 Z220.0 *
N011G00X160.0 Z180.0 *
N012G71U7.0R1.0 *
N013G71P014Q020U4.0W2.0F0.3S55 *
N014G00X40.0F0.15S58 *
N015G01W-40.0 *
N016    X 60.0W-30.0 *
N017    W-20.0 *
N018    X100.0W-10.0 *
N019    W-20.0 *
N020    X140.0W-20.0 *
N021G70P014Q020 *
```

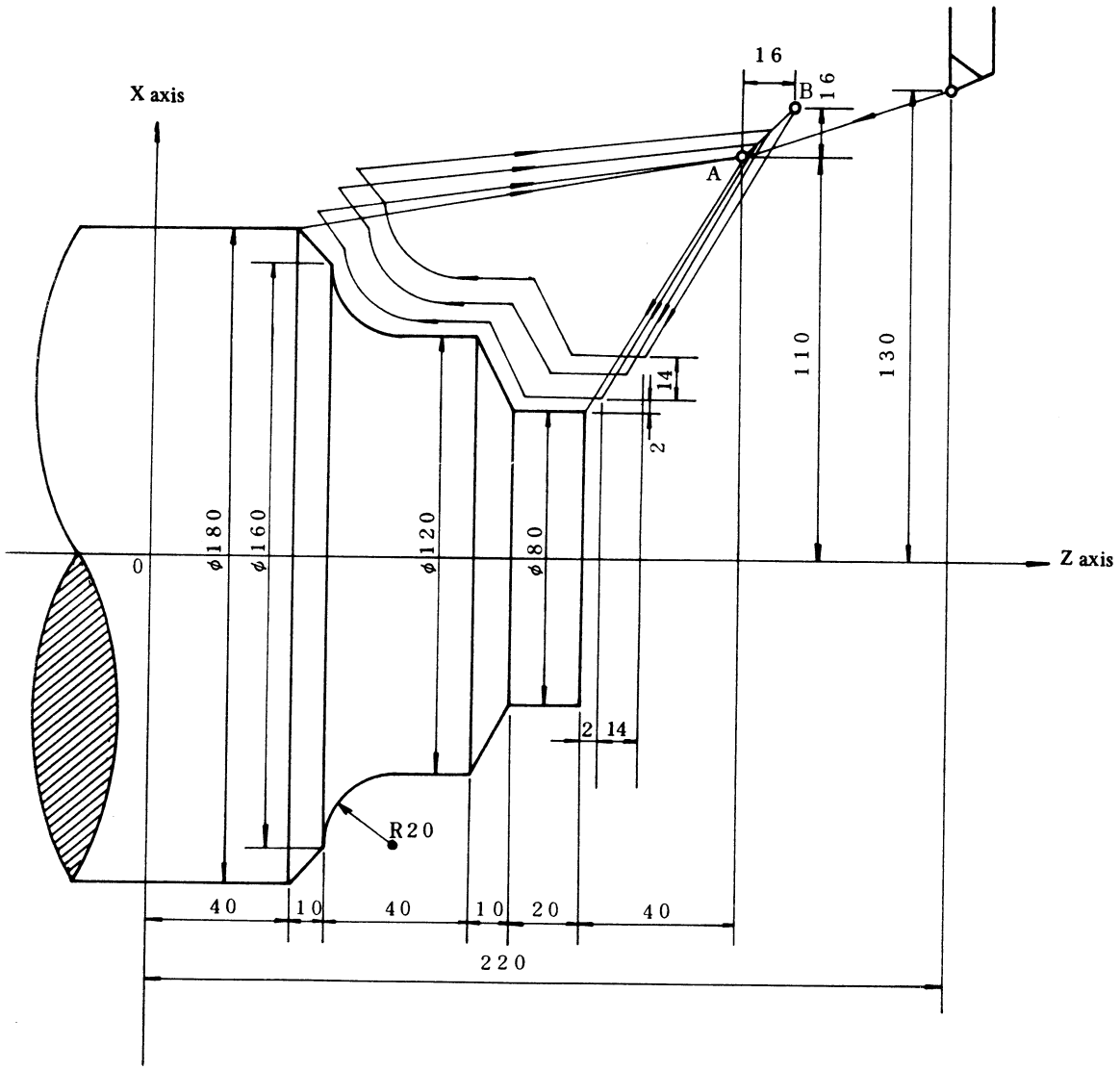
(Ex. 7.2.2) Example of programming by multiple repetitive cycle (G70, G72)



(Diameter designation, metric input)

```
N010G50X220.0 Z190.0 *
N011G00X176.0 Z132.0 *
N012G72W7.0R1.0 *
N013G72P014Q019U4.0 W2.0 F0.3S55 *
N014G00Z58.0S58 *
N015G01X120.0 W12.0 F0.15 *
N016    W10.0 *
N017    X80.0 W10.0 *
N018    W20.0 *
N019    X36.0 W22.0 *
N020G70P014Q019 *
```

(Ex. 7.2.3) Multiple Repetitive cycle

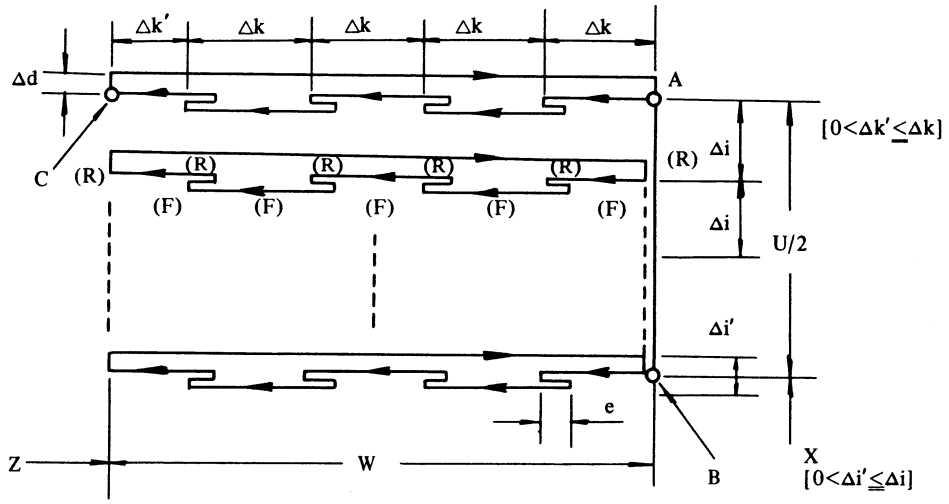


(Diameter designation, metric input)

```
N010 G50X260.0 Z220.0 *
N011 G00X220.0 Z160.0 *
N012 G73U14.0 W14.0 R3 *
N013 G73P014Q019U4.0 W2.0 F0.3 S0180 *
N014 G00X80.0 W-40.0 *
N015 G01W-20.0F0.15S0600 *
N016 X120.0 W-10.0 *
N017 W-20.0 S0400 *
N018 G02X160.0 W-20.0 R20.0 *
N019 G01X180.0 W-10.0 S0280 *
N020 G70P014Q019 *
```

7.2.5 Peck drilling in Z axis (G74)

The following command permits operation as shown in the figure below. Chip breaking is possible in this cycle as shown below. If X (U) and I are omitted, operation only in the Z axis results, to be used for drilling.



G74 R (e) *
G74 X (U) _ Z (W) _ P (Δi) Q (Δk) R (Δd) F (f) *

e : Return amount
This designation is modal and is not changed until the other value is designated. Also this value can be specified by the parameter (No. 87), and the parameter is changed by the program command.

X : X component of point B

U : Incremental amount from A to B

Z : Z component of point C

W : Incremental amount from A to C

Δi : Movement amount in X direction (without sign)

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

Δd : Relief amount of the tool at the cutting bottom. The sign of Δd is always plus (+). However, if address X (U) and Δi are omitted, the relief direction can be specified by the desired sign.

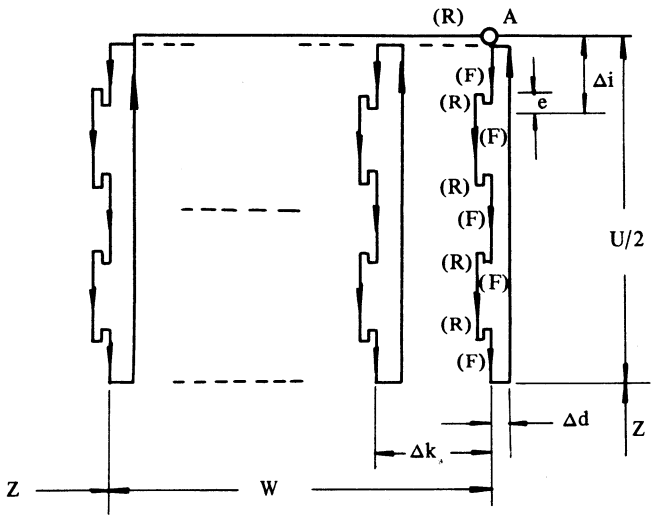
f : Feedrate

(Note 1) *While both e and Δd are specified by address R, the meanings of them are determined by the presence of address X (U).*

(Note 2) *The cycle machining is performed by G74 command with X (U) specification.*

7.2.6 Grooving in X axis (G75)

The following command permits operation as shown in the figure below. This is equivalent to G74 except that X is replaced by Z. Chip breaking is possible in this cycle, and grooving in X axis and peck drilling in X axis (In this case, Z, W, and Q are omitted) are possible.

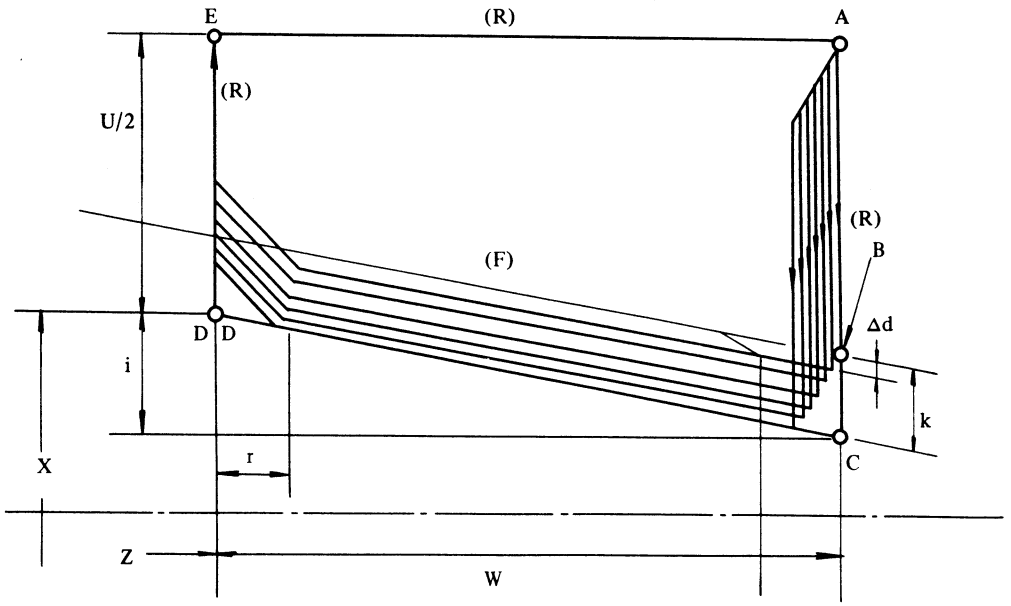


G75 R (e) *
G75 X (U) Z (W) P (Δi) Q (Δk) R (Δd) F (f) *

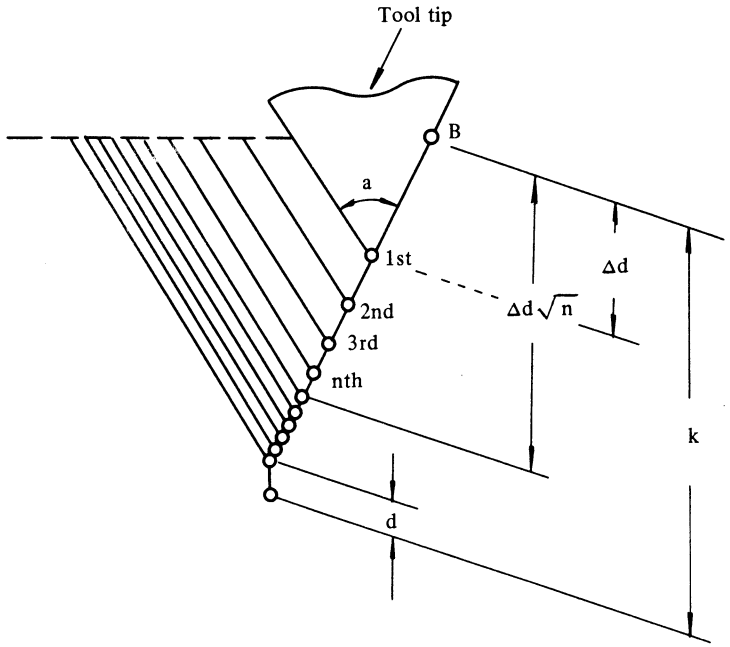
Both G74 and G75 are used for grooving and drilling, and permit the tool to relief automatically. Four symmetrical patterns are considered, respectively.

7.2.7 Thread cutting cycle (G76)

The thread cutting cycle as shown below is programmed by the G76 command.



(Detail of cutting)



G76 P(m)(r)(a) Q(Δdmin) R(d) *
G76 X(U)__ Z(W)__ R(i) P(k) Q(Δd) F(ℓ) *

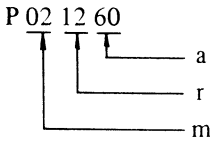
m : Repetitive count in finishing (1 to 99)
This designation is modal and is not changed until the other value is designated. Also this value can be specified by the parameter (No. 88), and the parameter is changed by the program command.

r : Chamfering amount
When the thread lead is expressed by ℓ, the value of r can be set from 0.0ℓ to 9.9ℓ in 0.1ℓ increment (2-digit number from 00 to 99).
This designation is modal and is not changed until the other value is designated. Also this value can be specified by the parameter (No. 22), the parameter is changed by the program command.

a : Angle of tool tip
One of six kinds of angle, 80°, 60°, 55°, 30°, 29°, and 0°, can be selected, and specified by 2-digit number.
This designation is modal and is not changed until the other value is designated. Also this value can be specified by the parameter (No. 89), the parameter is changed by the program command.

m, r, and a are specified by address P at the same time.

(Example) When m = 2, r = 1.2ℓ, and a = 60°, specify as shown below.



Δdmin : Minimum cutting depth
When the cutting depth of one cycle operation ($\Delta d\sqrt{n} - \Delta d\sqrt{n-1}$) becomes smaller than this limit, the cutting depth is clamped at this value. This designation is modal and is not changed until the other value is designated. Also this value can be specified by parameter (No. 90), the parameter is changed by the program command.

d : Finishing allowance
This designation is modal and is not changed until the other value is designated. Also this value can be specified by parameter (No. 91), the parameter is changed by the program command.

i : Difference of thread radius
If i = 0, ordinary straight thread cutting can be made.

k : Height of thread
This value is specified by the radius value in X axis direction.

Δd : Cutting depth in 1st cut (radius value)

ℓ : Lead of thread. (same as G32)

(Note 1) The meanings of the data specified by address P, Q, and R are determined by the presence of X (U) and Z (W).

(Note 2) The cycle machining is performed by G76 command with X (U) and Z (W) specification.

By using this cycle, one edge cutting is performed and the load on the tool tip is reduced. Making the cutting depth Δd for the first path, and $\Delta d \sqrt{n}$ for the nth pass, cutting amount per one cycle is held constant.

Four symmetrical patterns are considered corresponding to the sign of each address. The internal thread cutting is available.

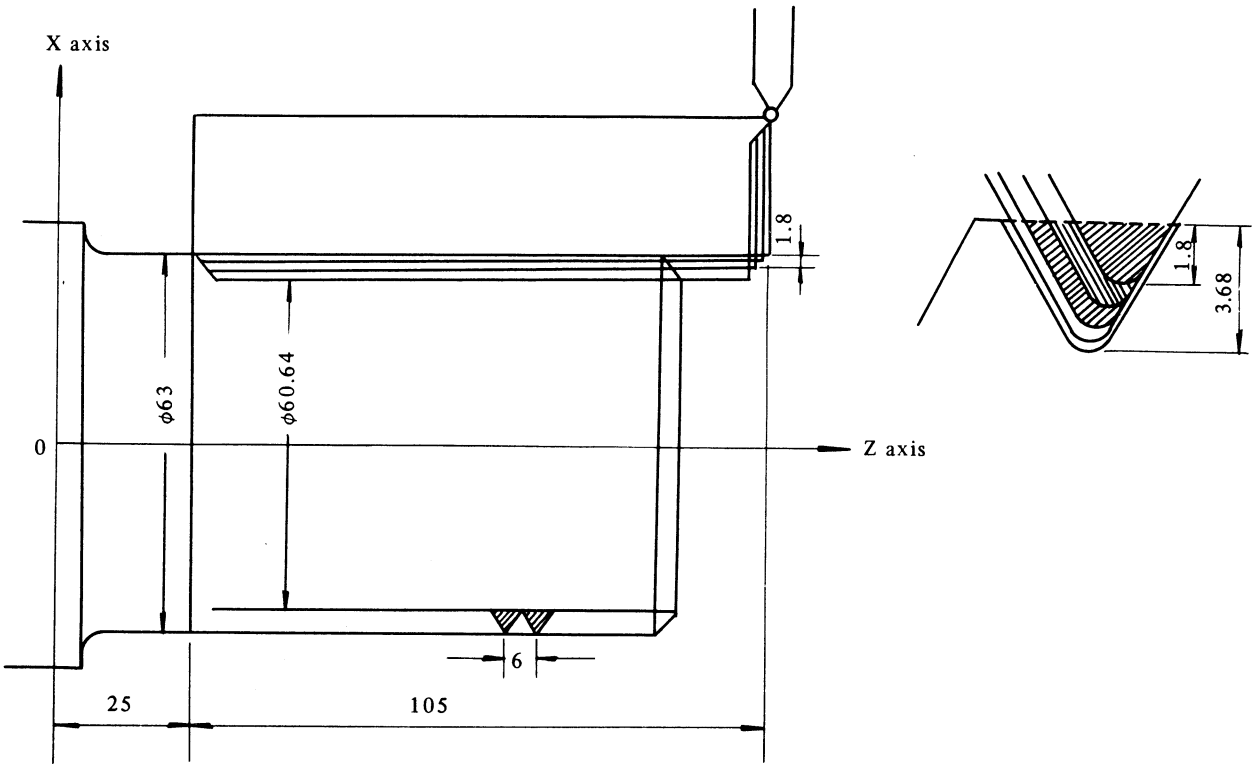
In the above figure, the feedrate between C and D is specified by address F, and in the other path, at rapid traverse.

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

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

R : Negative (determined by the direction of the tool path AC).

(Ex. 7.2.4) Example of programming by multiple repetitive cycle (G76)




```
G76 P011060 Q100 R200 *
G76 X60640  Z25000 P3680 Q1800 F6.0 *
```

(Note 3) *Notes on thread cutting are the same as those on G32 thread cutting and G92 thread cutting cycle.*

(Note 4) *The designation of chamfering is also effective for G92 thread cutting cycle.*

7.2.8 Notes on multiple repetitive cycle (G70 ~ G76)

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

(2) In the block which is specified by address P of G71, G72 or G73, G00 or G01 of 01 group should be commanded. If it is not commanded, P/S alarm (No. 65) is generated.

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

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

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

(a) One shot G code except for G04 (dwell)

(b) 01 group G code except for G00, G01, G02 and G03

(c) 06 group G code

(d) M98/M99

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

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

8. SPINDLE SPEED FUNCTION (S FUNCTION), TOOL FUNCTION (T FUNCTION), MISCELLANEOUS FUNCTION (M FUNCTION)

By commanding a numerical value following address S, T, M, a BCD signal and a strobe signal are transmitted to the NC machine tool and mainly used to control ON/OFF function of the machine.

The NC receives one of T, S and M codes in a block and signals are transmitted.

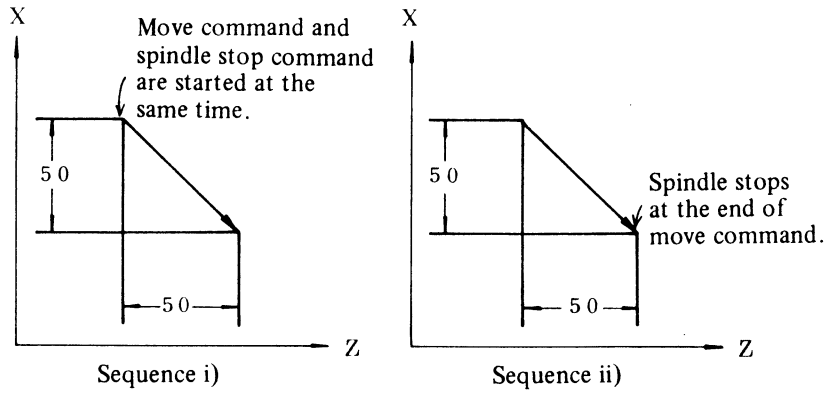
S codes are used for spindle speed control, T codes are used for tool change command, and M codes are used for ON/OFF of the various functions of the machine tool, etc. As for the application of these addresses and codes to functions, refer to the machine tool builder's manual.

When move command and S, T, M codes are commanded in the same block, the commands are executed either of two ways as follows.

(i) Simultaneous starting of move command and S, T, M functions commands.

(ii) Starting S, T, M functions commands upon completion of move command.

(Example) N1 G01U-100. 0 W50. 0 M05* (Spindle stop)



The selection of proper sequence depends on the machine tool builder's specifications. In certain cases, these sequences may be available together in an NC machine. Refer to the manual issued by the machine tool builder for detail.

8.1 Spindle Speed Function (S function)

The spindle speed is controlled by address S and a following 2-digit number. Refer to the manual issued by the machine tool builder for detail.

(Note) When a 4-digit S code is commanded in S 2-digit, the lower 2 digits are available.

8.2 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 and the voltage according to the spindle speed is transmitted to the spindle control section. The spindle speed is controlled so as to make the surface speed constant, by this function.

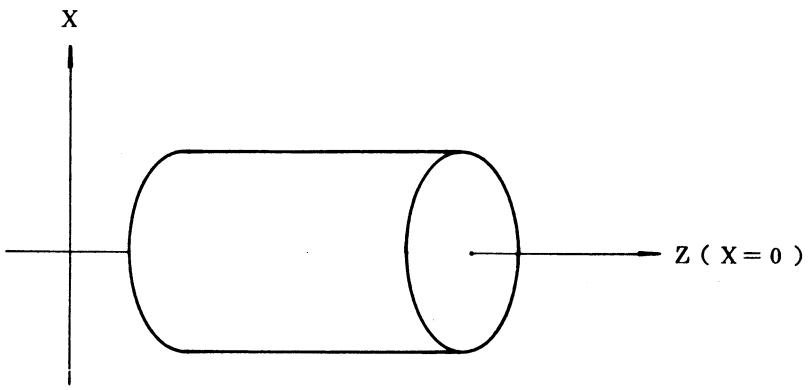
The unit of surface speed is as follow.

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

This surface speed unit varies with machine tool builders. 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 axis of rotation may be set to the Z axis (X = 0).



8.2.1 G50 commands a clamp of the maximum spindle speed

The figures following G50S specifies maximum spindle speed in constant surface speed control. Its unit is rpm.

G50 S____*

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

8.2.2 Rapid traverse (G00)

In a rapid traverse block with G00 command, surface speed is not calculated according to the change of tool position, but on the basis of the end position of the block because there is no cutting in rapid traverse.

(Note 1) When the power supply is turned on, the maximum spindle speed is not set, that is, it is not clamped.

(Note 2) Clamping is effective only when G96 is in use.

(Note 3) G50S0 means that the spindle speed is clamped to 0 rpm.*

(Note 4) The amount of S commanded in G96 mode is also kept in G97 mode and it is restored when returning to G96.

G96S50 (50m/min or 50 feet/min)*
G97S1000 (1000 rpm)*
G96X1000 (50m/min or 50 feet/min)*

(Note 5) The surface speed specified by constant surface speed control is for the program path. It is not for the position of tool offset amounts.

(Note 6) Even if a machine doesn't operate during machine lock, spindle speed for constant surface speed control is calculated according to the change of the program coordinate value of the X axis.

(Note 7) Constant surface speed control is effective in thread cutting mode. Therefore, it is better to cancel constant surface speed control by G97 in scroll thread cutting or taper thread cutting, so that the NC unit and servo system have no difficulty with the change of spindle speed.

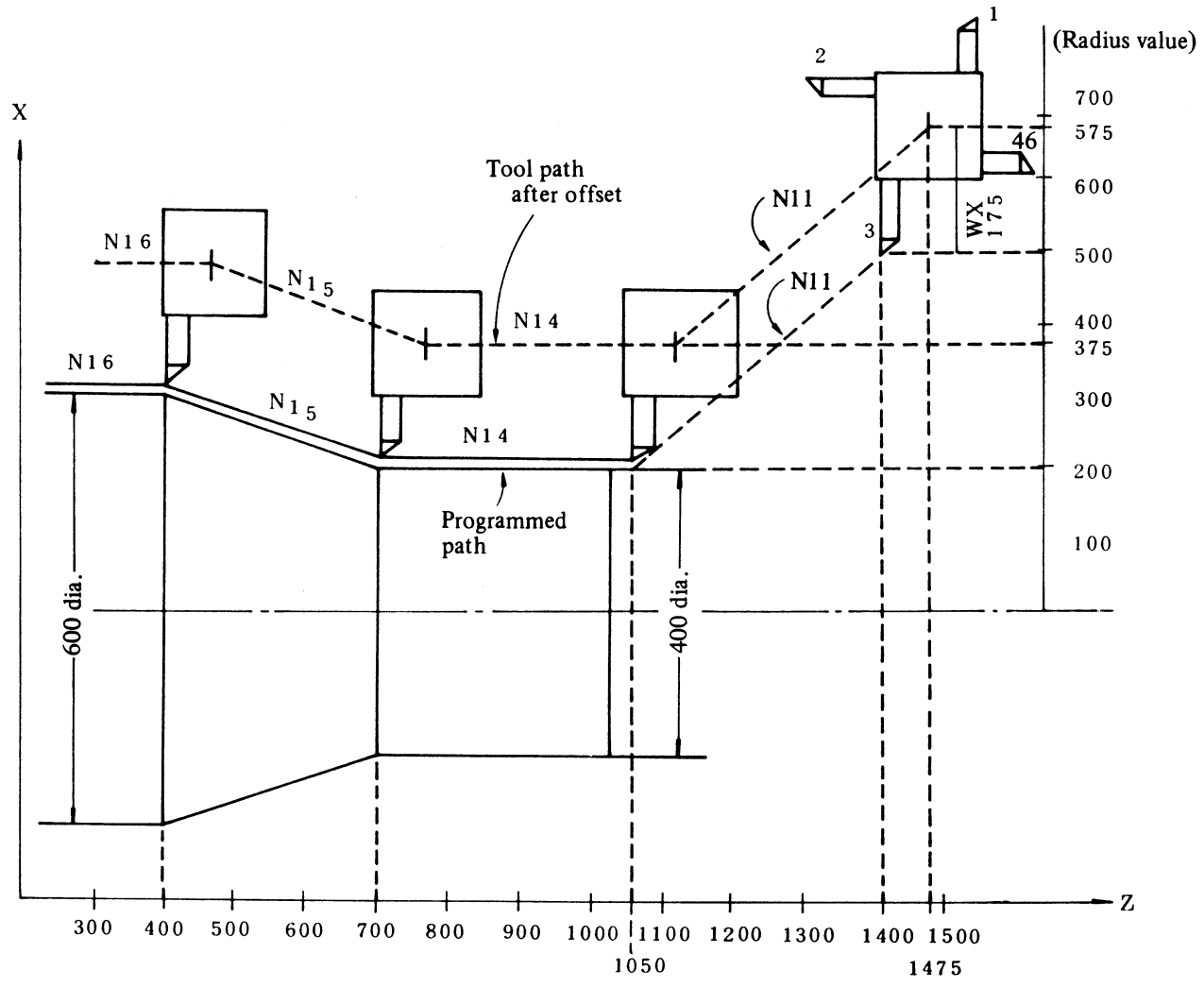
(Note 8) Although meaningless G96 (CSSC) and G98 (feed per minute) can be in effect at the same time.

(Note 9) When changing from G96 mode to G97 mode, the last spindle speed of G96 is used as the spindle speed for the G97 mode if it is not commanded in G97 mode.

N111G97S800 800 rpm*
⋮
N222G96S100 100 m/min*
⋮
N333G97 X rpm*

X is the spindle speed (X rpm) in the block previous to N333. When G97 mode is changed to G96, the value of S of G96 mode becomes effective. When S is not commanded, S=0 m/min (feet/min).

8.2.3 Programming example



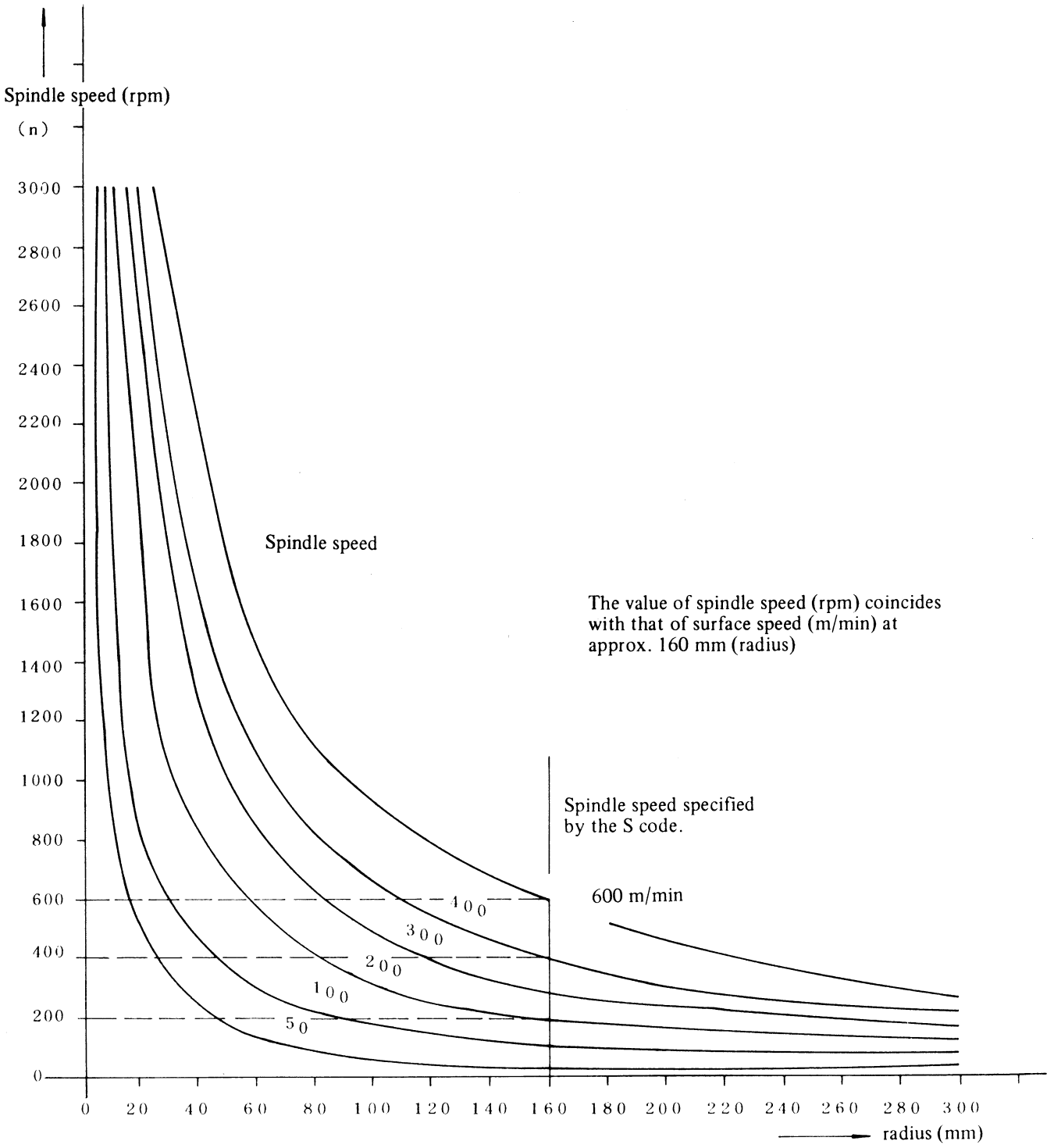
[Unit: mm]

(Diameter designation)

N 8 G00		X1000. Z1400.	*	
N 9			T0303 *	
N11	X400.	Z1050.	*	
N12 G50			S3000 *	(Designation of max. spindle speed)
N13 G96			S 200 *	(Surface speed 200 m/min)
N14 G01		Z 700.	F1000 *	
N15	X600.	Z 400.	*	
N16		Z	*	

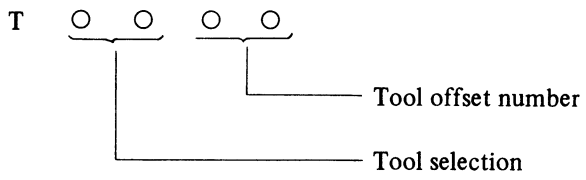
The NC calculates the spindle speed which is proportional to the specified surface speed at the position of the programmed coordinate value of the X axis, and it is not the value of X axis after offset when the offset is valid. At the end point N15 of the example above, the speed at 600 dia. which is not the turret center but the tool nose, is 200 m/min. If the X axis coordinate value is minus, the NC calculates using the absolute value.

8.2.4 Relation between spindle speed and surface speed



8.3 Tool Function (T function)

The value after T code indicates the desired tool. Also, a part of the value is used for the offset number which specifies the compensation amount of tool offset.



Refer to the manual issued by machine tool builder for correspondence between T-code and tool.
Example

N1 G00 X1000 Z1400
N2 T0313 . . . . (Select Tool No. 3 and Offset amount No. 13)
N3 X400 Z1050

This tool selection number is transmitted in BCD code with a strobe signal to the machine side.
(Note 1) Some machines set tool selection by 1-digit.

8.4 Miscellaneous Function (M function)

When a 2-digit figure is commanded following address M, a 2-digit BCD code signal and a strobe signal are transmitted. These signals are used for ON/OFF control of machine function. One M code can be commanded in one block. When two or more M codes are commanded, only the last one is effective. Selection of M codes for functions depends on the machine tool builder.

Following M codes are used for special meaning.

8.4.1 M02, M30: End of program

i) It shows the end of main program. It is necessary for registration from tape to memory.
ii) Cycle operation is stopped and the NC unit is cleared. (It differs according to the machine tool builders.)

8.4.2 M00: Program stop

Cycle operation is stopped after a block containing M00 is executed. When the program is stopped, all existing modal information remains unchanged, as in single block operation. The cycle operation can be restarted by commanding an NC start. (It differs according to the machine tool builders.)

8.4.3 M01: Optional stop

Similar to the M00, cycle operation is stopped after a block which contains M01 is executed. This code is effective only when the Optional Stop switch has been turned ON on the machine control panel.

8.4.4 M98: Calling of subprogram

This code is used to enter a subprogram. Refer to the section on subprogram control for details.

8.4.5 M99: End of subprogram

This code shows the end of a subprogram. Executing M99 take the control back to the main program. Refer to the section on subprogram control for details.

(Note 1) If there is a block following M00, M01, M02, M30, it is not read into the buffer.

(Note 2) In M98 code or M99 code, a code signal and a strobe signal are not transmitted.

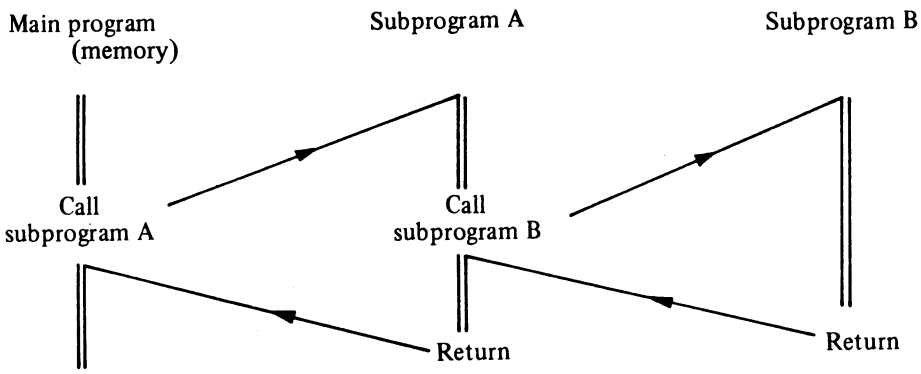
(Note 3) M codes except M98, M99 are processed by the machine. Nothing is processed only by the NC unit. Refer to the manual issued by the machine tool builder.

9. SUBPROGRAM**

When a program contains certain fixed sequences or frequently repeated patterns, these sequence or pattern may be entered into the memory as a subprogram. This can simplify programming.

The subprogram can be called out in AUTO mode. A subprogram can call for another subprogram.

While the subprogram called for by the main program is regarded as a one loop subprogram call, two loop subprograms can be performed, as is shown below.



A call command can call a subprogram repeatedly. A call command can command the repeating of a subprogram up to 999 times.

9.1 Preparation of Subprogram

A subprogram is prepared in the following format:

```
O (:) XXXX *
..................*
..................*
      .
      .
      .
      .
M99 *
```

At the top of a subprogram, a subprogram No. which represents the subprogram is placed after 'O' (EIA) or ':' (ISO). Subprogram end command 'M99' need not be commanded in a block by itself.

(Example)

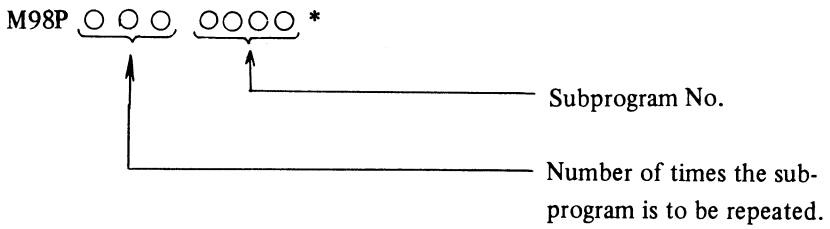
```
X..................M99*
```

For entering of subprograms into the memory, refer to section IV.10.13 and 10.14.

(Note 1) For compatibility of NC tape, 'Nxxxx' is also used as a subprogram No. In this case, the sequence number is used for subprogram number.

9.2 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:



When repetitive count is omitted, the subprogram is repeated once.

(Example) M98P51002*

This command is read "Call the subprogram (subprogram number 1002) five times."

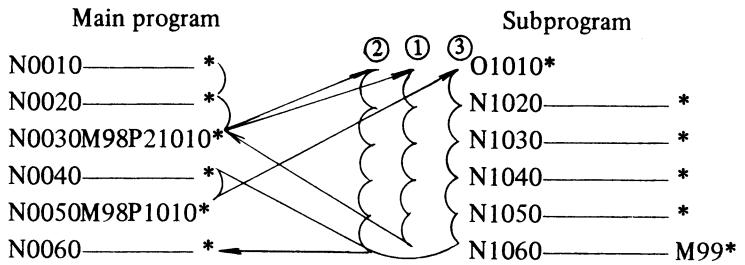
The subprogram call command (M98P___) and a move command can be commanded in the same block.

(Example) X1000 M98P 1200*

In this case, after completing moving in X-axis direction, the subprogram (subprogram number 1200) is called once.

(Example)

In the main program with subprogram call command, the executing sequence is as follows:



When the subprogram is called for by another subprogram, the sequence executed is the same as in the above example.

(Note 1) M98 and M99 signals are not transmitted to the machine side.

(Note 2) If the subprogram number specified by address P can not be found, the alarm (No. 78) is generated.

(Note 3) In MDI operation, the subprogram call command 'M98P ____' cannot be used. In this case, prepare the main program in EDIT mode as follows:*

*O xxxx**
*M98P xxxx**
*M02**

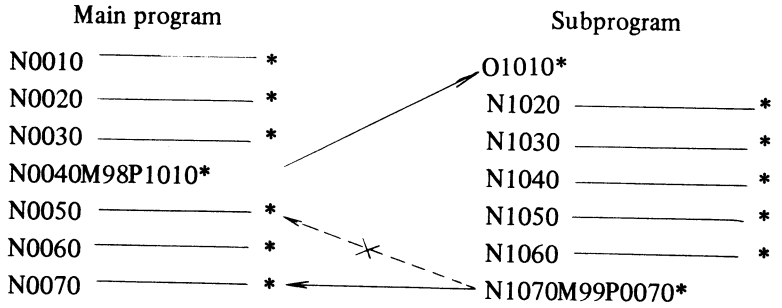
And then, execute it in cycle operation.

9.3 Special Uses

The following special uses are available.

9.3.1

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 subprograms calling block, last block, but to the block with the sequence number specified by the address P.



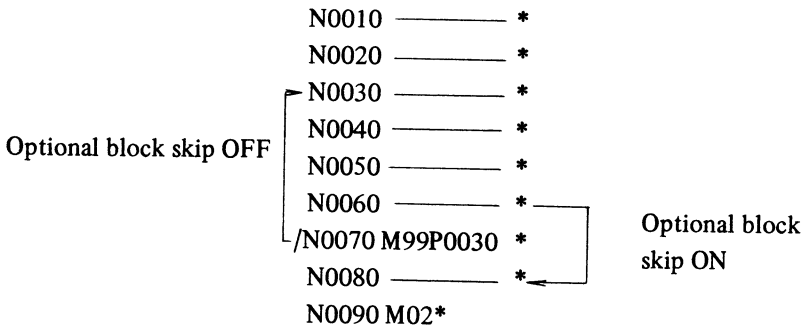
9.3.2

If the M99 command is executed in the main program, the control returns to the start of the main program.

For example, if inserting a '/M99*' block in the proper position of the main program and if the optional block skip is off, then M99 is executed, and the control returns to the start of the main program and executes the program again.

If the optional block skip is turned on, the M99 is omitted and control goes to the next block.

If the '/M99Pn*' has been inserted, the control does necessarily return to the start, but it returns to the block whose sequence number is 'n'.

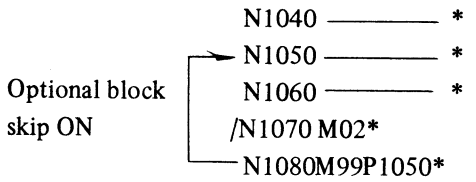


9.3.3

It is possible to execute a subprogram from the start by searching for it via the MDI keyboard as well as main program. (Refer to IV. 10.15)

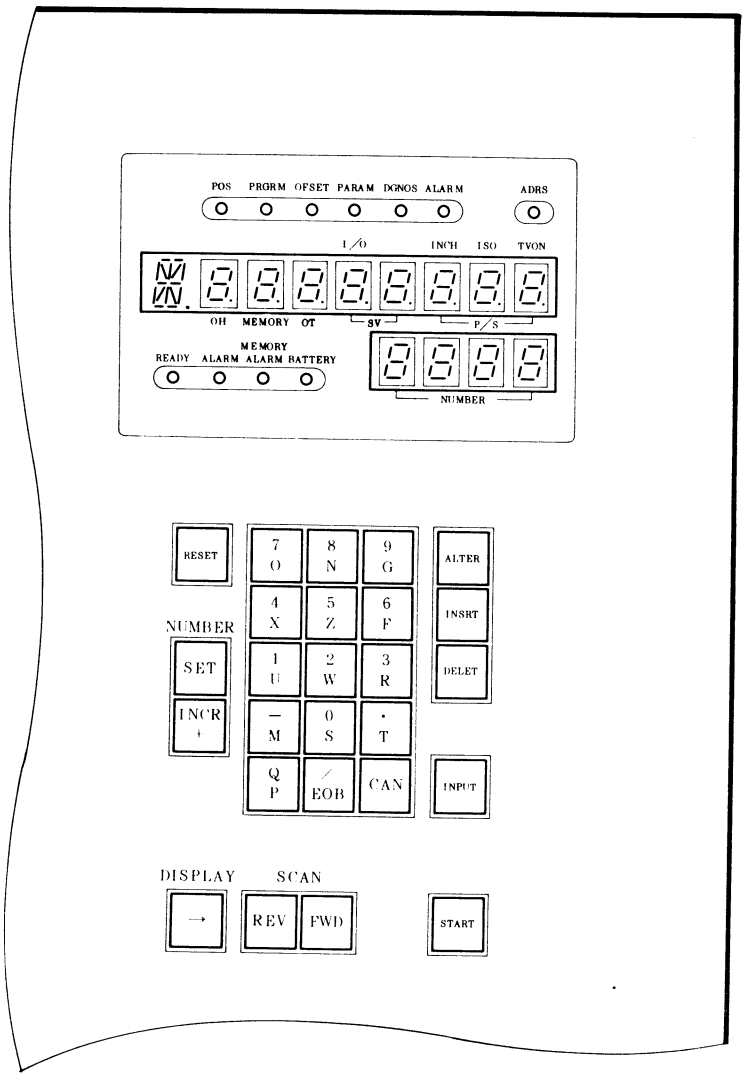
In this case, if the M99 command is executed, the control returns to the top of the subprogram and execution is repeated. If the 'M99Pn*' is executed, the control returns to the block whose sequence number is 'n' and execution is repeated.

In the above operation, if you want to stop the execution, insert the '/M02*' or the '/M30*' at the proper position. When the optional block skip switch is turned off, the above command is executed and the program is terminated. In the example below, execution will continue while the optional block skip is on and it will stop when the optional block skip is turned off.

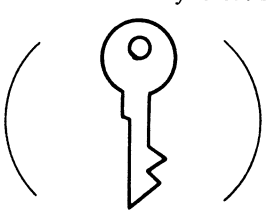

N1040 ——— *
N1050 ——— *
Optional block
skip ON
N1060 ——— *
/N1070 M02*
N1080M99P1050*

10. DISPLAY AND OPERATION ON THE MDI & DPL PANEL

The MDI & DPL panel consists of the switches, buttons, lamps, etc. as show in the photo.

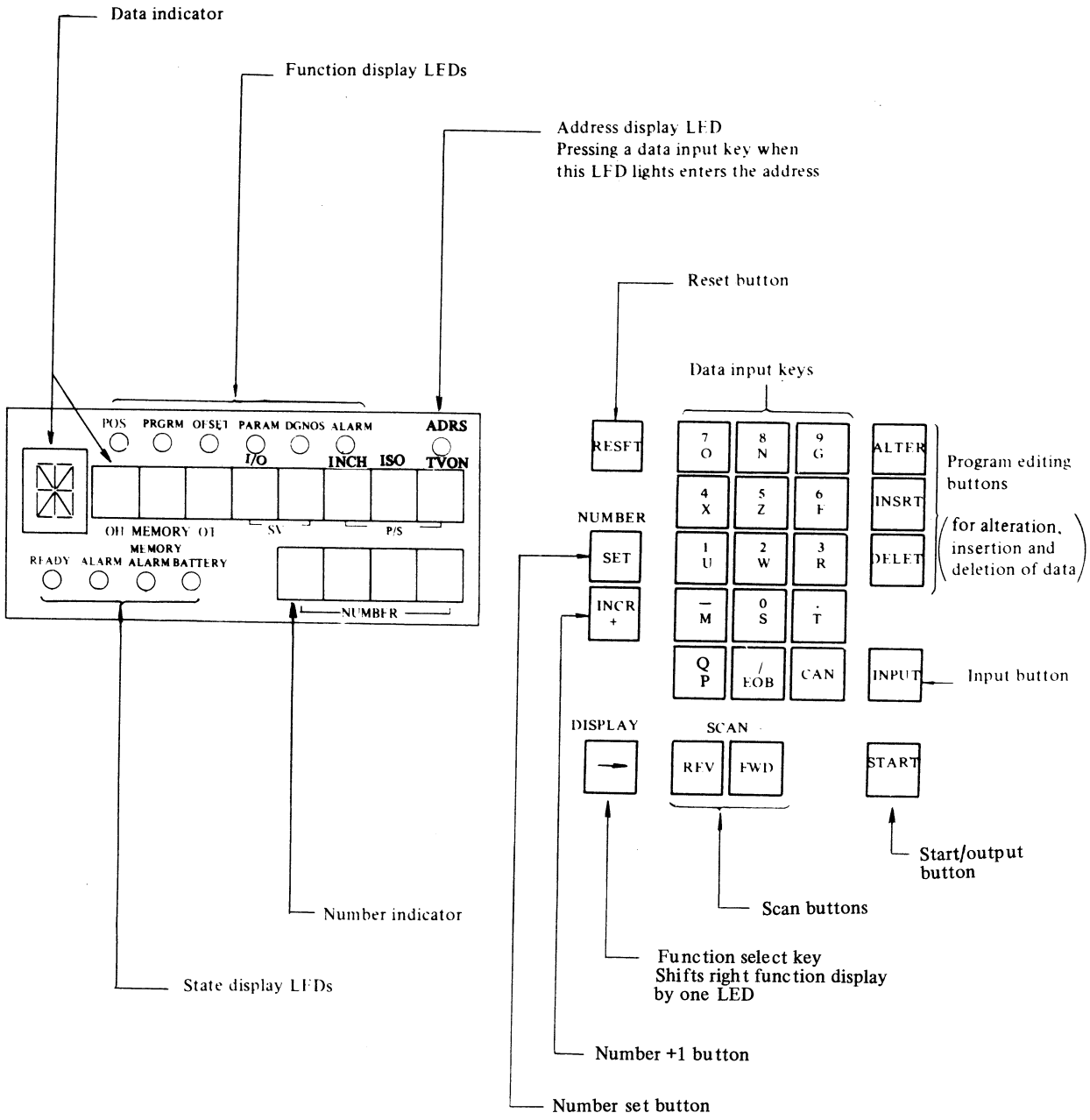


The LOCK key can be provided for operator's panel by machine manufacturer.
If the LOCK key is not released, operations can not be performed.



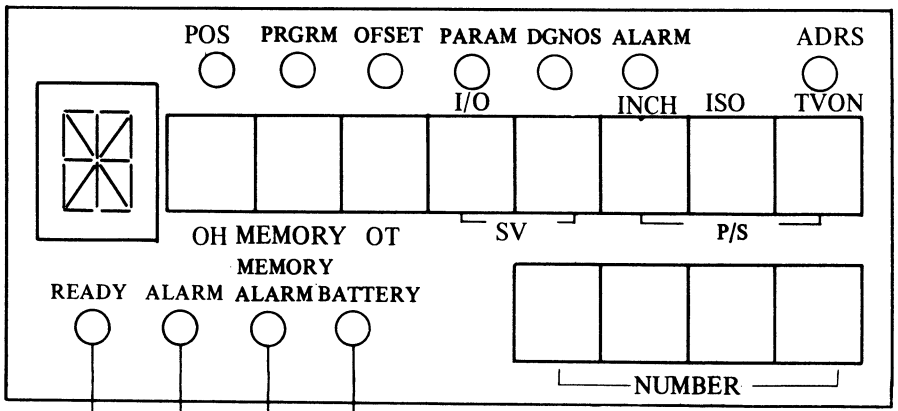
..... The operation with the mark are possible only when the LOCK key is released.

Even if the operation have this mark, the operation to display on the indicator can be performed.


Data indicator
Function display LEDs
Address display LED
Pressing a data input key when
this LED lights enters the address
Reset button
Data input keys
POS PRGRM OFSET PARAM DGNOS ALARM ADRS
I/O INCH ISO TVON
OH MEMORY OT
SV P/S
MEMORY
READY ALARM ALARM BATTERY
NUMBER
RESET
7 O
8 N
9 G
4 X
5 Z
6 F
1 U
2 W
3 R
— M
0 S
. T
Q P
/ EOB
CAN
ALTER
INSRT
DELET
Program editing buttons
for alteration, insertion and deletion of data
NUMBER
SET
INCR +
INPUT
Input button
DISPLAY
SCAN
REV
FWD
START
Start/output button
Scan buttons
Function select key
Shifts right function display
by one LED
Number +1 button
Number set button
Number indicator
State display LEDs

10.1 State Display LEDs

The following four states can be displayed by LEDs.



BATTERY LED
Lights when the battery voltage drops below a certain level. This battery protects information in the memory when the power is cut. Replace the battery when this LED lights.

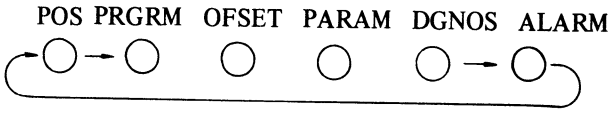
MEMORY ALARM LED
Lights when an error occurs in memories such as ROM, RAM, and C-MOS.

ALARM LED
Lights when any alarm of OH, MEMORY, OT, SV, and P/S is detected. Lighting the ALARM LED displays the contents of the alarm on the data indicator.

READY LED
Lights when both the control equipment and the servo system are ready to operate. When this LED extinguishes, no NC operation is possible.

(Note 1) When a servo system made by other than FANUC is in use, the READY LED sometimes lights without checking that the servo system is ready to operate.

10.2 Function Display LEDs



Display the type of data displayed on the data indicator or the function of the data entered. Pressing the function selector key DISPLAY shifts right the lighting position one at a time. (Next to ALARM LED, it is shifted to POS LED) Following data input and display are available, corresponding to each LED.

(1) POS LED

Display of the current position and display of modal address data.

(2) PRGRM LED

Editing and display of a program in memory in the EDIT mode and input and display of MDI data in the MDI mode.

(3) OFSET LED

Input and display of a tool offset amount

(4) PARAM LED

Input and display of a parameter.

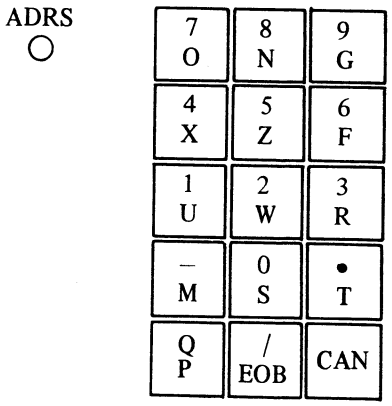
(5) DGNOS LED

Display of diagnostic data (DI, DO).

(6) ALARM LED

Display of an alarm number.

10.3 Address Display LED and Data Input Keys



Data input keys, as in the left figure, are used as numeric keys and as address keys.

The ADRS LED displays whether a number or an address is to be entered.

When [Q P] or [/ EOB] is pushed once, P or EOB code is entered respectively. When twice, Q or / (slash code) is entered.

When the ADRS LED lights, an address is entred; when not, a number. The NC automatically turns on and off the ADRS LED depending on various conditions (function display LED and so on).

Example: Entering 'X123' on the MDI panel.

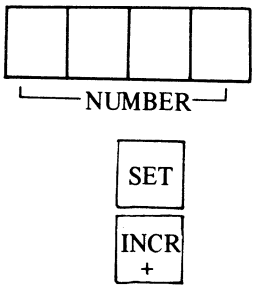
① Selecting PRG in the MDI mode lights the ADRS LED, making the key-in of an address possible.

② Pressing the [4 X] key keys in X, extinguish the ADRS LED, and makes the key-in of a number possible.

③ Pressing the [1 U], [2 W], and [3 R] keys in this order keys in '123'.

④ Pressing the Input key enters 'X123' and lights the ADRS LED.

10.4 Number Indicator



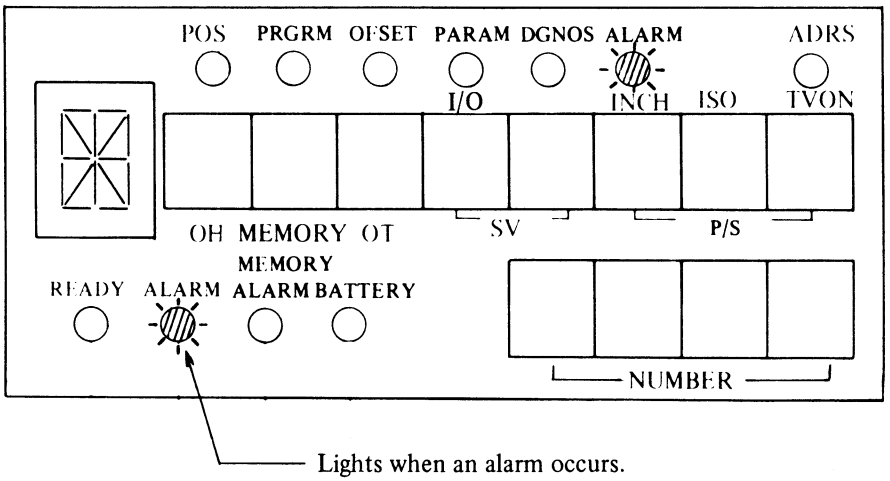
The number indicator displays a sequence number, an offset number, parameter number, a diagnostic data number, and a group number for addresses G and S.

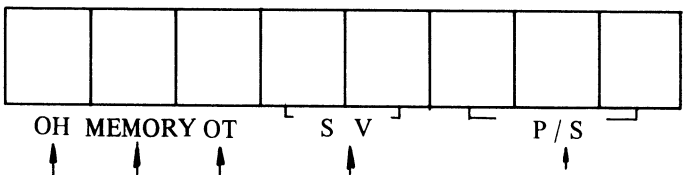
State	Displayed information
OFSET LED lights	Offset number
PARAM LED lights	Parameter number
DGNOS LED lights	Diagnostic data number
G code is displayed at other than program editing time	Group number of G code
POS LED lights and address S is displayed.	Group number of address S
PRGRM LED lights in EDIT mode	Sequence number scanned last in memory
Program number search	Program number searched for
Program comes out to the start of program or memory	Program number of the program
Sequence number search	Sequence number searched for
Other	Sequence number or program number executed last

An offset number, a parameter number, a diagnostic number, and group numbers for address G and S can be set and changed. Pressing the INCR + key adds 1 to the number. Press the SET key extinguishes number on the number indicator. Data input keys are pressed to key-in a desired data number into the number indicator. Pressing the INPUT key enters the key-in data into the number indicator and displays the data corresponding to it on the data indicator.

10.5 Alarm Display

When an alarm occurs and the ALARM LED lights, lighting the ALARM LED by pressing the DISPLAY key a required number of times displays the type and number of the alarm on the data indicator.





Displays the error number in 3-digit value (000-999) when a program error occurs.

Displays the error number in 2-digit value (01-99) when a servo alarm occurs.

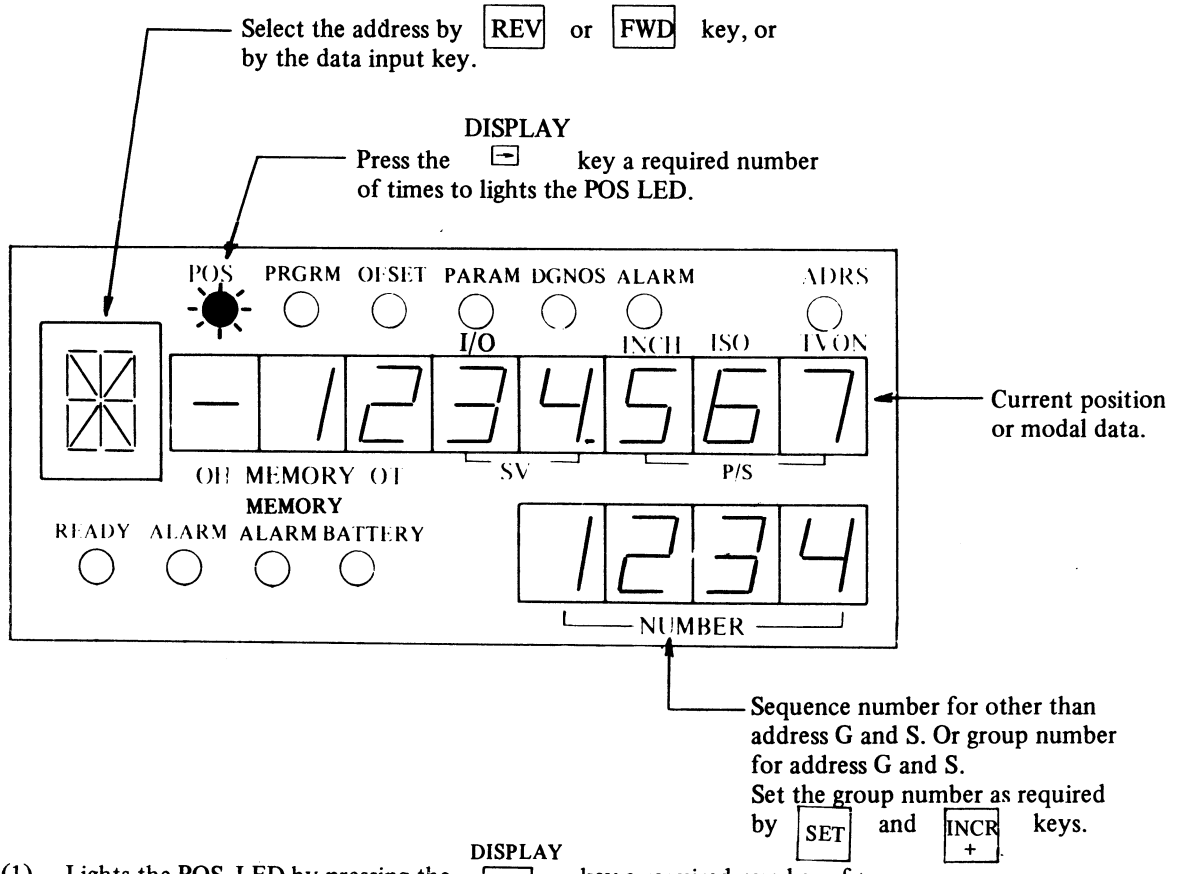
Displays the error number in 1-digit value (1-9) when an overtravel alarm occurs.

Displays the error number in 1-digit value (1-9) when the memory fails (the MEMORY ALARM LED also lights).

Displays the error number in 1-digit value (1-9) when an overheat alarm occurs.

Any part of the data indicator not corresponding to the generated alarm is blank.
Refer to Appendix 4 for the meanings of each alarm number.

10.6 Display of the Current Position and Modal Data



(1) Lights the POS LED by pressing the DISPLAY → key a required number of times.

(2) Select an address to be displayed in one of the following two ways available:

(a) Press the data input key for a desired address (the ADRS LED does not extinguish, so pressing one data input key after another changes the address.)

(b) Press the REV and FWD keys to scan addresses. Pressing the REV key switches the address as in

O → N → G → X → Z → U → W → R → F → M → S → T → P → Q

Pressing the REV key switches the address as in

O ← N ← G ← X ← Z ← U ← W ← R ← F ← M ← S ← T ← P ← Q

(3) This operation displays data in the following table correspondingly on the data indicator and the number indicator.

Address	On Data Indicator	On Number Indicator
X	X coordinate value in coordinate system set by G50	Number of sequence executed last
Z	Z coordinate value in coordinate system set by G50	
U	X relative coordinate value (Note 1)	
W	Z relative coordinate value (Note 1)	
O, N, F M, S, T	Value of current state	
R, Q	Commanded value in execution. Blank in other case.	
P	Commanded value during execution Blank in other cases.	
G	Modal G code of each group corresponding to group number	Group number of G code
	Group 00: Blank	00
	Group 01: G00, G01, G02, G03, G32, G90, G92, G94	01
	Group 02: G96, G97	02
	Group 03: Special G90, G91 only (Note 2)	03
	Group 04: G68, G69	04
	Group 05: G98, G99	05
	Group 06: G20, G21	06
S	Spindle revolution information of each group corresponding to group number	Group number of S code
	Per-minute number of revolutions (rpm) or code command	0
	Surface speed (meters/min. or feet/min.)	1
	Maximum allowable number of revolutions (rpm)	2
	Actual spindle speed (rpm)	3

(Note 1) *The coordinate values in relative coordinate system can be set to zero any time regardless of program.*

(Note 2) *G codes of other groups are written in standard G code.*
For address G and S, a group number to be displayed can be set on the number indicator.
(See Section 10.4)

10.7 Display of a Command Value During Execution

The command value of one block being executed during automatic operation can be displayed.

(1) Press the DISPLAY [→] key as many a time as required to light the PRGRM LED.

(2) As in Section 4.6 (2), select the desired address by the [REV] and [FWD] keys or the data input key.

(3) This operation displays the command value on the data indicator. The blank is displayed in the case of the address not commanded.

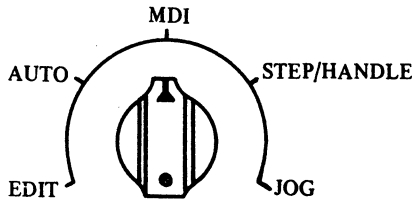
For other than address G, the number indicator displays the sequence number executed last. For address G, the indicator displays the group number, which can also be set by the [SET] and [INCR +] keys.

10.8 Operation by the MDI Panel

One block of command can be entered from the MDI & DPL panel for execution.

(1) Executing X10.5W200.5*, for example.

(a) Set the mode select switch to MDI.

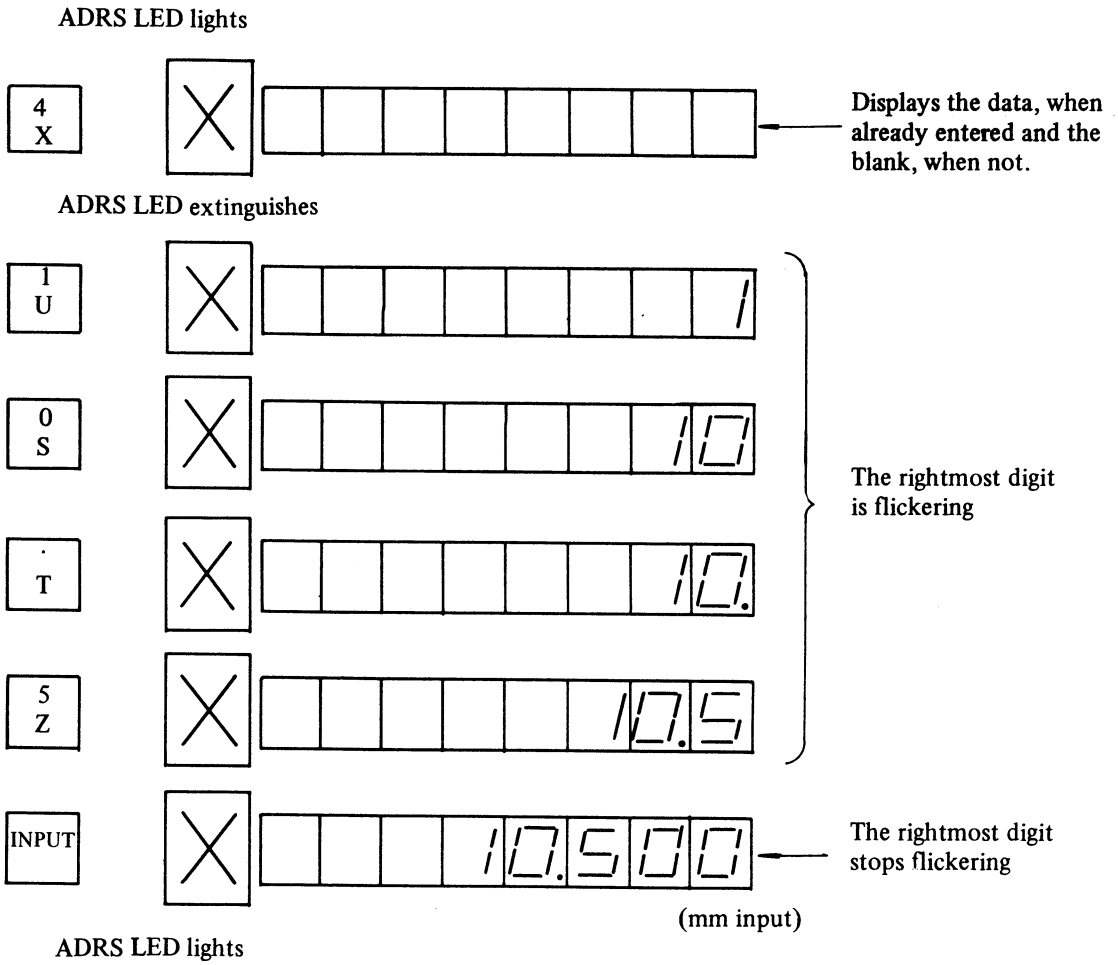


(b) Select PRGRM.

(c) After checking that the ADRS LED lights, press the [4 X] key. The ADRS LED will extinguish.

(d) Press the data input keys to key in '10.5'.

(e) Press the INPUT button. 'X10.5' will be entered and the ADRS lamp lights.

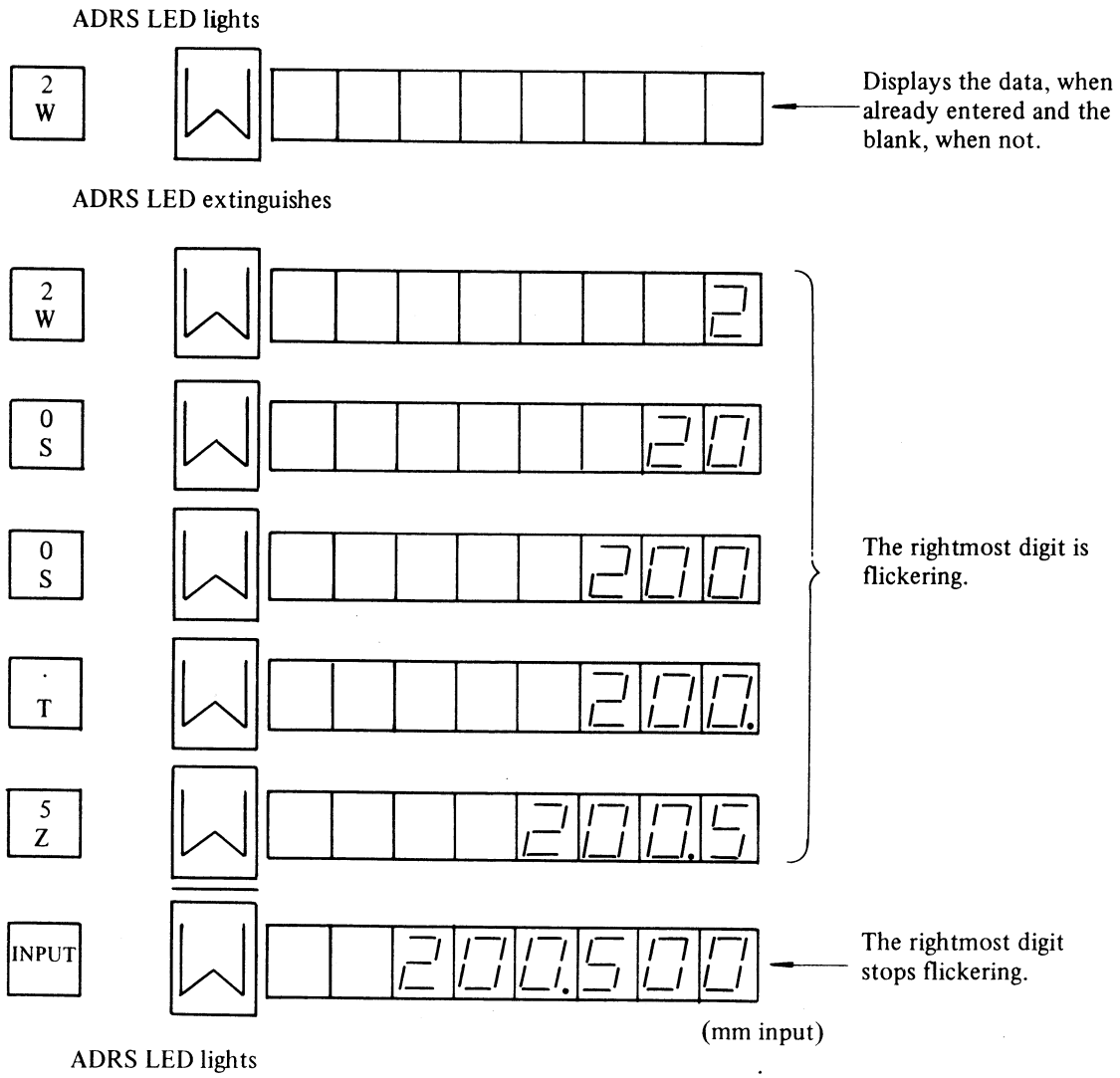


When you are aware of keying error before pressing the INPUT button, press the CAN key, then key in X again and the correct number (pressing the CAN key lights the ADRS LED). When you are aware of keying error after pressing the INPUT button, key in X again and the correct number.

(f) Press the [2 W] key. W will be keyed in and the ADRS LED extinguishes.

(g) Press the data input keys to key in '200.5'.

(h) Press the [INPUT] button. 'W200.5' will be entered and the ADRS LED lights.



When keying in a wrong number, do as in the case of X.

(i) Select addresses X and W by the [REV] and [FWD] keys to check the input data. If wrong, repeat the steps of (c) to (h).

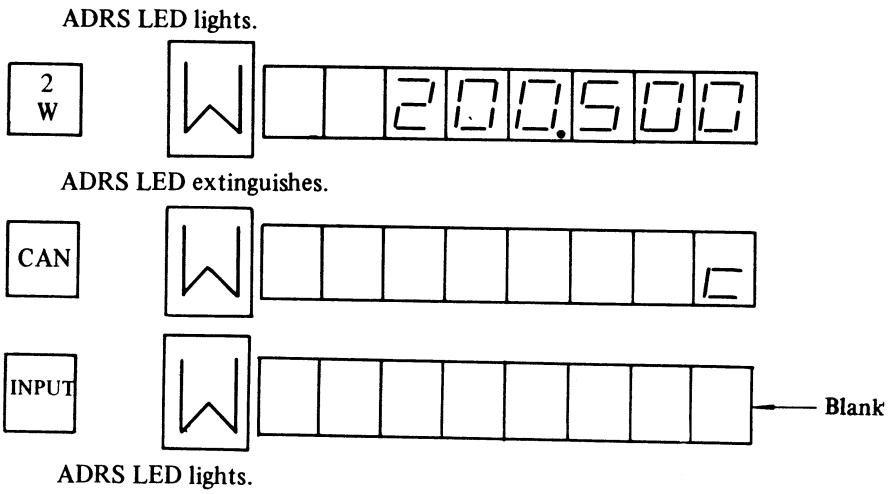
(j) Press the [START] button. Press the cycle start button on the control panel, depending on the machine tool.

(2) Canceling W200.5 in the preceding example before pressing the [START] button.

(a) After checking that the ADRS LED lights, press the W key. W will be keyed in and the ADRS LED extinguishes.

(b) Press the [CAN] key.

(c) Press the [INPUT] button. The data entered into address W will be canceled and the ADRS LED lights.



(3) Checking the entered data

Select the address for the data to be checked. The data indicator displays the data, if entered and is blank, if not. When address G is selected, the number indicator displays the group number. Changing the number on the number indicator by the SET and INCR + buttons displays the G code of the group corresponding to that number on the data indicator. For an address other than G, the number indicator displays the sequence number executed last. Pressing the REV and FWD keys to scan addresses selects a desired address.

Successively pressing the FWD key switches the address as in

O → N → G → X → Z → U → W → R → F → M → S → T → P → Q

Successively pressing the REV key switches the address as in

O ← N ← G ← X ← Z ← U ← W ← R ← F ← M ← S ← T ← P ← Q

(Note 1) Pressing a data input key also selects the address. But in this way the ADRS LED extinguishes, so it is difficalt to switch the address successively. Pressing the CAN key twice after the address key lights the ADRS LED.

10.9 Reset

The RESET button is pressed, generally to remove an alarm state. The NC enters into the following state by pressing this button. For details, refer to the appendix on reset states when the power is on, or in reset state. side.

Before reset	After reset
Move commands are being executed.	The tool slows down to stop, the remaining move commands disappear.
M, S, or T code is being sent out.	The sequence being sent out stops. Refer to the manual of the machine builder as to that happens at the machine side.
The MDI operation data has been entered.	All entered data is canceled.

10.10 Resetting A Relative Coordinate Value

(1) Select POS by the DISPLAY → button to light the POS LED.

(2) Select address U or W to be reset, as in item 10.6 (2).

(3) Pressing the CAN key reset to zero the X axis (address U) or Z axis (address W) relative coordinate value. The relative coordinate value can be reset in an automatic-operation stopped state, in a feed hold state or in a reset state.

10.11 Setting and Display of a Tool Offset Amount.

10.11.1 Input of an absolute amount

(1) Press the DISPLAY → button as many a time as required to light the SET LED.

(2) Set the offset number on the number indicator in one of the following ways:

(a) Pressing the INCR + button adds +1 to the offset number.

(b)

(i) After confirming that the ADRS LED lights, press the SET button. An blank is displayed on the number indicator and the ADRS LED extinguishes.

(ii) Key in the offset number to be set by the data input keys.

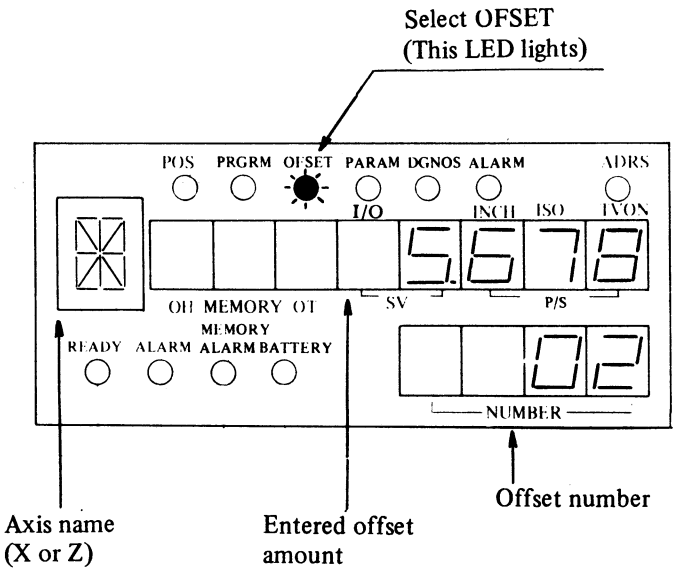
(iii) Press the INPUT button. The keyed-in number will be entered into the number indicator and the ADRS LED lights.

(3) Press the 4 X key (for X axis) or the 5 Z key (for Z axis) for the address to be set. The data indicator will display the currently set offset amount with X or Z and the ADRS LED extinguishes.

(4) Key in the offset amount (the decimal point will do) by the data input keys.

(5) Press the INPUT button. The offset amount will be entered and the ADRS LED lights.

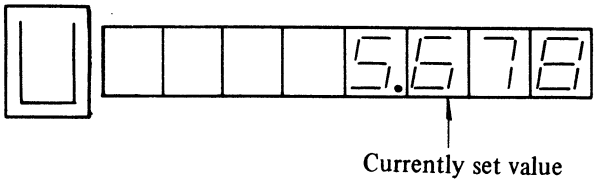
(6) The entered offset amount is displayed as follows.



10.11.2 Input of an incremental amount

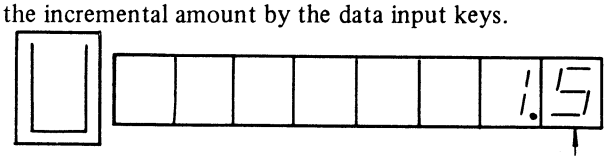
(1) Set the offset number on the number indicator as in 10.11.1 (2).

(2) After confirming that the ADRS LED lights, press the [1 U] key (for X axis) or the [2 W] key (for Z axis) for the address to be set. The data indicator will display the currently set offset amount with U or W and the ADRS LED extinguishes.



Currently set value

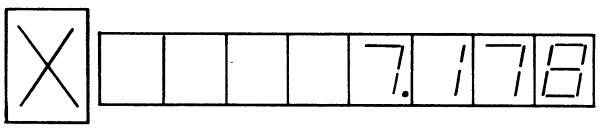
(3) Key in the incremental amount by the data input keys.



The rightmost digit is flickering.

(4) Press the [INPUT] button. The incremental amount will be entered and added to the current offset amount and the ADRS LED lights.

(5) The address changes to X or Z and the newly set offset amount is displayed.



(Note) The offset amount, when changed during automatic operation, does not become valid until the T code specifiying that next offset number is commanded.

10.11.3 Display of an offset amount

(1) Select OFSET to light the OFSET LED.

(2) Set the desired offset number on the number indicator by the [SET] and [INCR +] buttons.

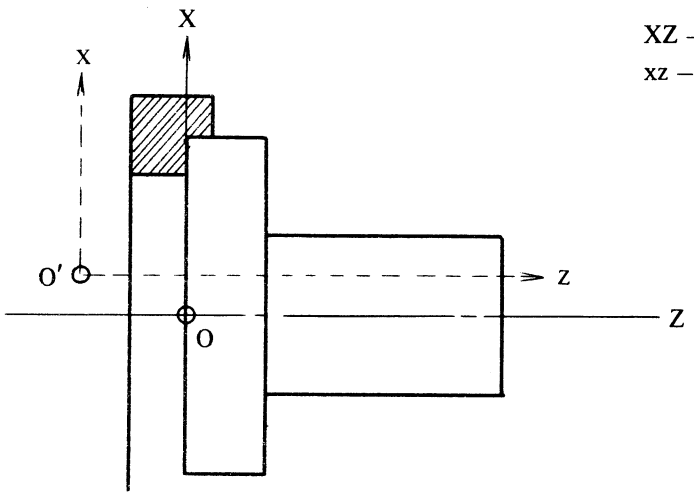
(3) Press the [REV] and [FWD] keys to scan addresses for a desired address. Pressing either the [REV] key or the [FWD] key switches between X and Z.

(4) The data indicator displays the offset amount for the selected number and axis.

10.11.4 Setting of work coordinate system shift amount

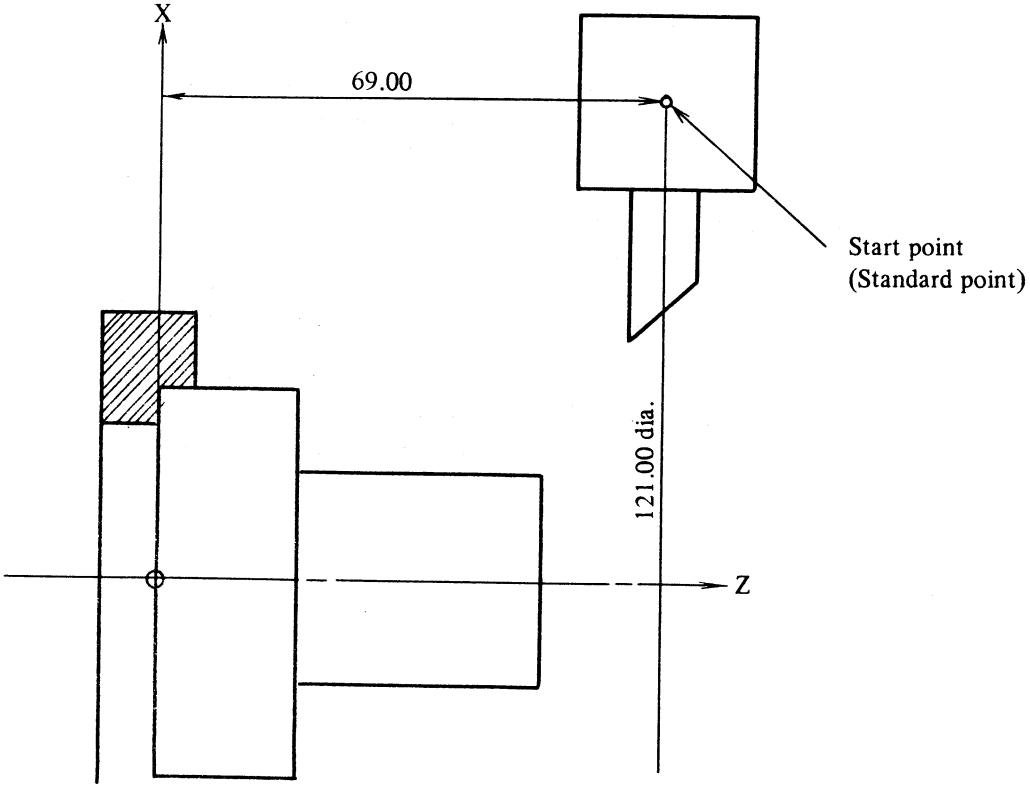
If the work coordinate system set by G50 (G92) command or automatic coordinate system setting function is different from the coordinate system in programming, the coordinate system set in advance can be shifted.

Set the shift amount to the offset memory of offset number 00. Setting procedure of shift amount is same as that in tool offset amount.



XZ – O : Coordinate system in programming
xz – O' : Currently set work coordinate system

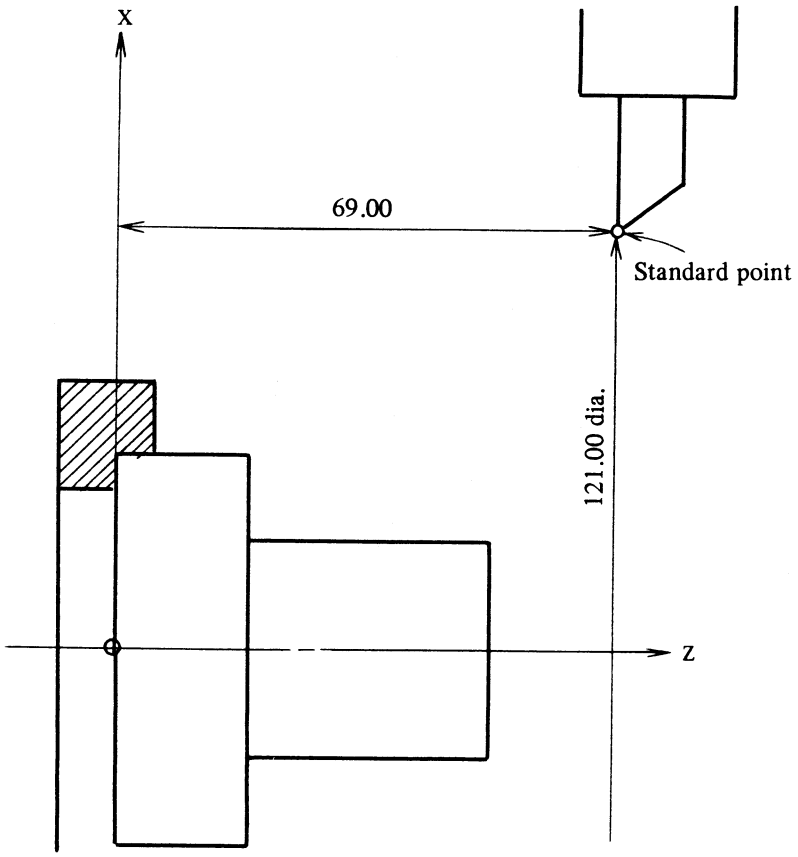
Set the shift amount, O' to O, to the offset memory with number 00.



When the actual standard point is at the position of 121 mm (diameter value) in X axis and 69 mm in Z axis even though the following G50 (G92) command is executed,

G50 (G92) X 120.0 Z 70.0 *

the desired work coordinate system can be set by shifting the current coordinate system with 1.0 mm in X axis and –1.0 mm in Z axis.



G50 X 120.0 Z70.0 * (Diameter programming)

If the standard point is at the tool tip as shown in the above figure, the coordinate system can be shifted and the desired work coordinate system can be set. The shift amount is 1.0 mm in X axis and −1.0 mm in Z axis.

At actual setting of shift amount, the method in item 10.11.5 is simple and convenient "Direct measured value input for work coordinate system shift".

(Note 1) The work coordinate system is shifted immediately after setting the shift amount.

(Note 2) After setting the shift amount, if G50 command is programmed, the shift amount becomes ineffective.

(Example) If the following command is specified, the coordinate values of the current standard point is X = 100.0 and Z = 80.0 being irrespective of the shift amount previously set.

*G50 X 100.0 Z 80.0 **

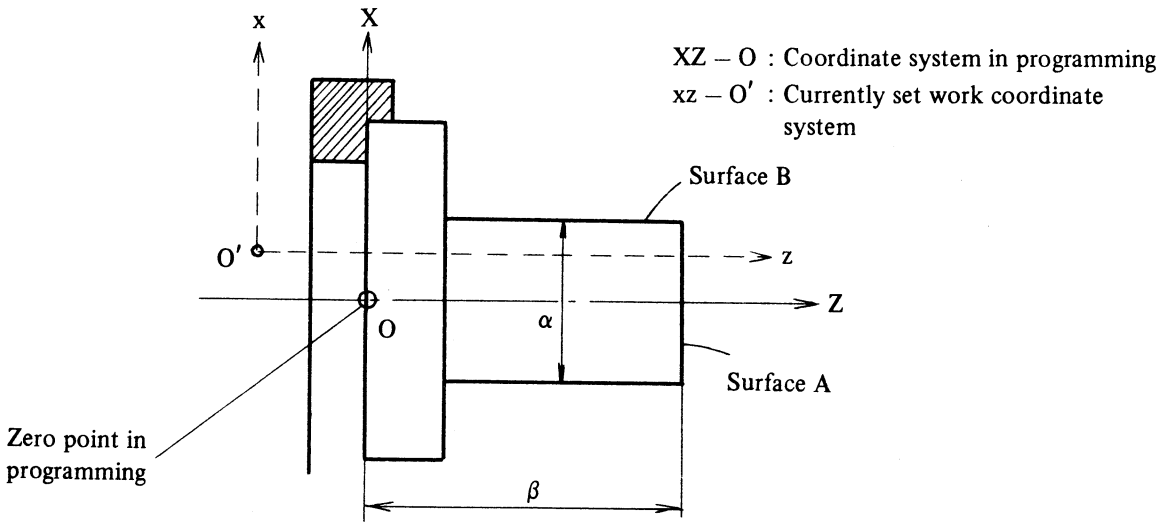
(Note 3) If the automatic coordinate system setting is performed by manual reference point return after shift amount setting, the coordinate system is shifted instantly.

(Note 4) The work coordinate system shift becomes effective or not by parameter setting (WSFT in parameter number 12).

(Note 5) The shift amount in X axis is programmed in diameter/radius designation depending on that in program.

10.11.5 Direct measured value input for work coordinate system shift

When the work coordinate system set by G50 command or the automatic coordinate system setting function is different from the coordinate system in programming, the coordinate system can be shifted by storing the measured distance directly instead of the shift amount as follows.

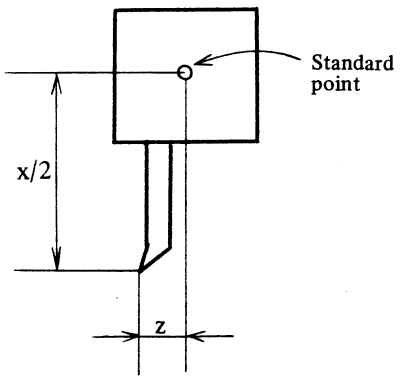


(1) Cut the workpiece along surface A using the standard tool in manual operation.
(2) Release the tool along X axis without Z axis movement and stop the spindle rotation.
(3) Measure the distance "β" in the figure above and input it to the offset memory with offset number 100 for Z axis.
(4) Cut the workpiece along surface B in manual operation.
(5) Release the tool along Z axis without X axis movement and stop the spindle rotation.
(6) Measure the diameter "α" and input it to the offset memory with offset number 100 for X axis.

The shift amount, O′ to O, is automatically set to the offset memory with offset number 00, and the work coordinate system is shifted immediately.

If the offset amount of the standard tool is zero, the work coordinate system in which the coordinate values of the tool tip is X = 0.0 and Z = 0.0 at positioning the tip of the standard tool to the zero point is set.

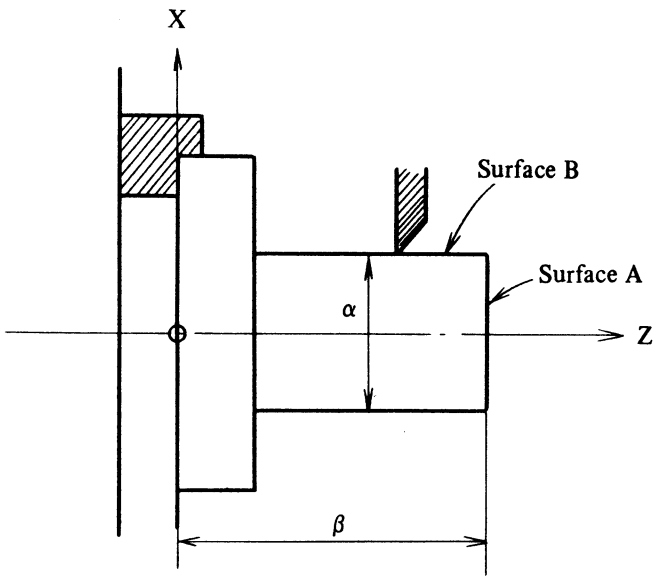
(Note 1) If the tool offset amount is set as shown in the figure below and the tool offset function is effective and the direct measured value input for the work coordinate system shift is performed, the work coordinate system in which the coordinate values of the standard point is X = 0.0 and Z = 0.0 at positioning the standard point to the zero point is set.



(Note 2) The distance "α" should be set in diameter value.

10.11.6 Direct measured value input for tool offset

The following convenient method is available in setting the offset amount that the difference between the standard point, such as the tip of the standard tool or the turret center, and the tip of the tool actually used. Here, the work coordinate system should be set in advance.



(1) Cut the workpiece along surface A using the tool in actual machining.
(2) Release the tool along X axis without Z axis movement and stop the spindle rotation.
(3) Measure the distance "β" between the surface A and the zero point of the work coordinate system, and set it at the offset number which is added one hundred to the offset number desired to set the offset amount.
(4) Cut the workpiece along surface B in manual mode.
(5) Release the tool along Z axis without X axis movement and stop the spindle rotation.
(6) Measure the diameter "α" and set it to the offset number explained in Step (3).

By the above procedure, the offset amounts are set to the desired offset number.

If the coordinate value of the tool at the surface B is 105.0, the measured distance is 104.0 and it is set to the offset number 103, the offset amount 1.0 is automatically set to the offset number 03.

(Note 1) The direct measured value input for tool offset becomes effective or not depending on the parameter setting (DOFSI of parameter number 12).

(Note 2) The distance "α" should be set in diameter value.

10.11.7 Counter input of offset amount

In offset amount setting, if the INPUT button is pushed after only setting the address (X, U, Z or W) without numerical value input, the present relative coordinate value set as an offset amount corresponding to the address. It is convenient to use by the following procedure.

(1) Position the tip of the standard tool to the standard point.

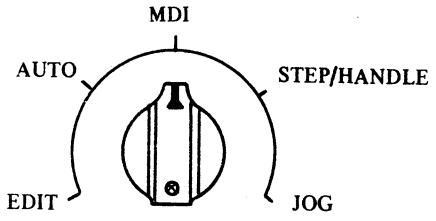
(2) Reset the relative coordinate values (U or W).

(3) Set the tool used for actual machining and position the tool tip to the standard point.

(4) [4 X] [INPUT] and [5 Z] [INPUT] after setting the offset number. The offset amount is set to the desired offset number. Accordingly, the offset amount between the standard tool and the actual used tool is indicated as the relative coordinate values.

10.12 Setting of SETTING Parameters

(1) Set the mode select switch to MDI.

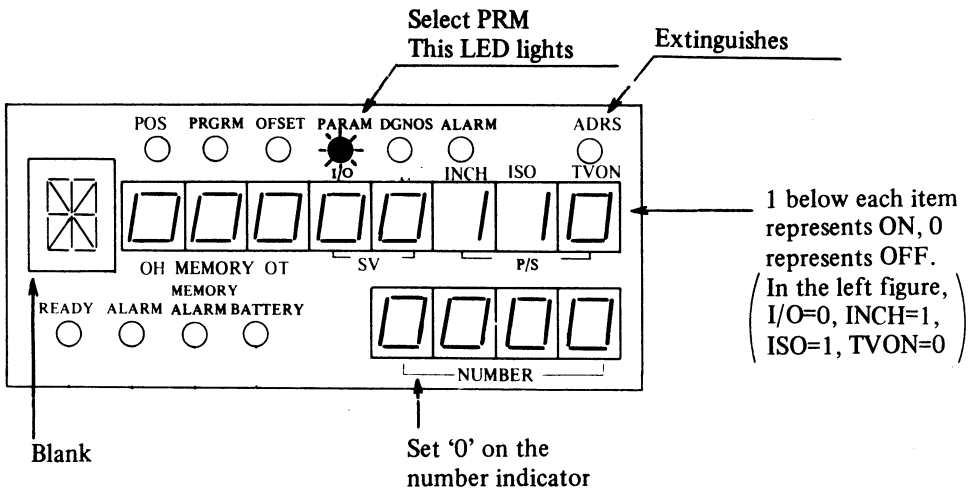


(2) Press the [DISPLAY →] button as many a time as required to light the PARAM LED. The ADRS LED extinguishes by selecting PARAM.

(3) Press the [SET] and [INCR +] buttons to set to zero the number on the number indicator. (the [SET] button is effective also when the ADRS LED lights.)

(4) Key in 1 or 0 in the order of DRN, INCH, ISO, and TVON.

(5) Press the [INPUT] button. Each setting parameter will be set and displayed.



(a) I/O

1: I/O device is selected by parameters "NFED", "ASR33" and "STP2" of parameter No. 14, and "BRATE 1" of parameter No. 69. Refer to appendix 6 for details.

0: I/O device is selected by parameters "NFED", "ASR33" and "STP2" of parameter No. 5 and "BRATE 0" of parameter No. 68. Refer to appendix 6 for details.

(b) INCH

Input system in programming is

1: Inch input

0: Metric input

(c) ISO

Output code is

1: ISO code

0: EIA code

(d) TVON

TV check is

1: effective

0: ineffective

10.13 Registration of Program into Memory ()

(1) Registration by MDI operation.

(a) Set the MODE SELECT switch to EDIT mode.

(b) Select the PRGRM and light the PRGRM LED.

(c) Key-in the address O.

(d) Key-in the program number to be registered.

(e) Push the INSERT key.

The program number can be registered by the above operation. After that, key-in the program and push the INSERT key word by word. (For details, refer to item 11.5 Insertion of Word)

(2) Registration by paper tape

(a) Set MODE SELECT switch to EDIT or AUTO mode.

(b) Set the switch of punch panel to MODE 3 and connect the signal cable of portable tape reader.

(c) Set "1" to the setting parameter "I/O".

(d) Set the following parameters.

Parameter No. 14	"ASR33" = 0, "STP2" = 1
Parameter No. 69	"BRATE 1" = 4800

(e) Set a program tape to the portable tape reader.

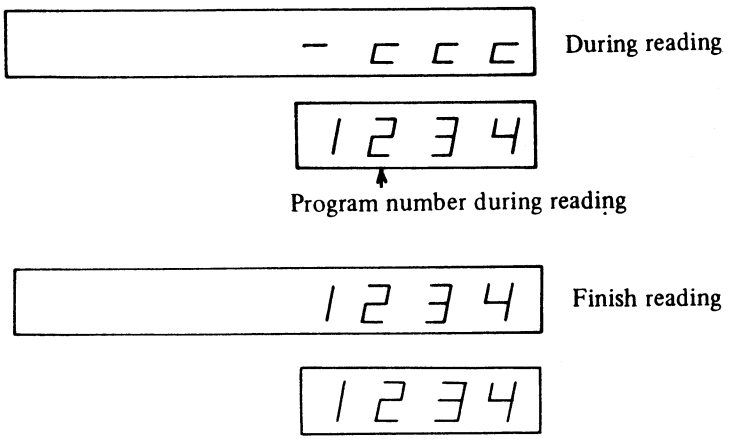
(f) Select the PRGRM and light the PRGRM LED.

(g) Input a program number, when the program tape does not have a program number or when to change the program number. (When not, it is not necessary.)

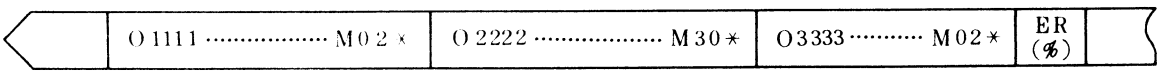
(i) Select address O.

(ii) Input program number.

(h) Push INPUT key.



10.14 Many Programs Registration from Tape ( )



The operation procedure follows in item 10.13 Registration of Program into Memory. Programs on the tape are entered into the memory, until ER code appears.

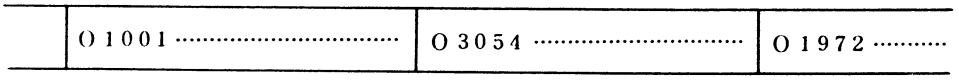
The program number is put as following manners.

(1) When it is not entered by MDI operation.
  (a) Number following address O, or when it is omitted, following first address N is set as the program number.
  (b) If they are not exist, the number added +1 to the previous program number is set.

(2) When it is entered by MDI operation, the number following address O is ignored, and the specified number is set at the first program. The program number of the following programs are added +1 to the previous number.

10.15 Program Number Search

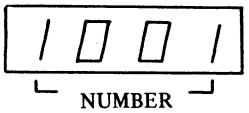
The program in memory can be searched.



→ Direction of program number search

(1) Method 1

(a) Set MODE SELECT switch to EDIT mode or AUTO mode.
(b) Select PRGRM and light PRGRM LED.
(c) Push address O.
(d) Input program number to be search.
(e) Push FWD.
(f) When the program number search is finished, the searched program number is displayed in the number indicator.



(2) Method 2

(a) Set MODE SELECT switch to EDIT mode or AUTO mode.
(b) Select PRGRM and light PRGRM LED.
(c) Push the address O.
(d) Push FWD.

When keeping pushing FWD, program numbers are displayed one by one.

After all of the entered program numbers are displayed, the display returns to the top.

(3) Method 3

(a) Set MODE SELECT switch to AUTO mode.
(b) Reset the NC.
(c) Set the signal at the machine side to 01 to 15 (for details, refer to the manual issued by machine tool builder).
(d) Push CYCLE START button.

The program number, from 0001 to 0015 corresponding to the signal at the machine side is searched and the cycle operation is started by the operation, from step (a) to (d).

(Note 1) If the signal at the machine side is '00', program number search is not performed.
(Note 2) If the program corresponding to the signal at the machine side is not registered in the memory, an alarm (No. 59) occurs.
(Note 3) Reset state is that the LED, which lights during cycle operation, extinguishes. (For details, refer to the manual issued by machine tool builder).

10.16 Deletion of Program ()

The program in memory can be deleted.

(1) Set MODE SELECT switch to EDIT mode.
(2) Select PRGRM and light the PRGRM LED.
(3) Push the address O.
(4) Input program number.
(5) Push DELET key, and the program with the input number is deleted.

10.17 Deletion of all Programs ()

All programs in memory can be deleted.

(1) Set MODE SELECT switch to the EDIT mode.
(2) Select PRGRM and light the PRGRM LED.
(3) Push the address O.
(4) Input [−] [9] [9] [9] [9] DELET

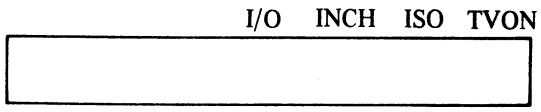
10.18 Punch of Program

All programs in memory can be punched.

(1) Turn on the tape punch unit.
(2) Set the switch of punch panel to MODE 3 and connect the signal cable of portable tape reader.
(3) Set "1" to the setting parameter "I/O".

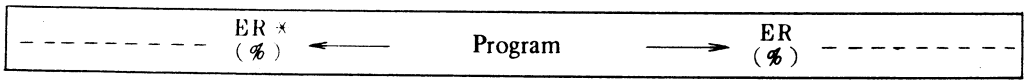
(4) Set the following parameters according to the tape punch unit used.
Parameter No. 14 "ASR33" and "STP2"
Parameter No. 69 "BRATE 1"

(5) Set the punch code by the setting parameter "ISO".



Set ISO code output to 1.
Set EIA code output to 0.

(6) Set MODE SELECT switch to EDIT mode.
(7) Select the PRGRM and light the PRGRM LED.
(8) Key-in the address O.
(9) Key-in the program number.
(10) The program to be input is punched by pushing START .



Feed (3 feet) Feed (3 feet)

(Note 1) *The space code for TV check are automatically punched.*

(Note 2) *When the program is punched in ISO code,* (LF) (CR) (CR) *are punched.*

..... (LF) (CR) (CR)

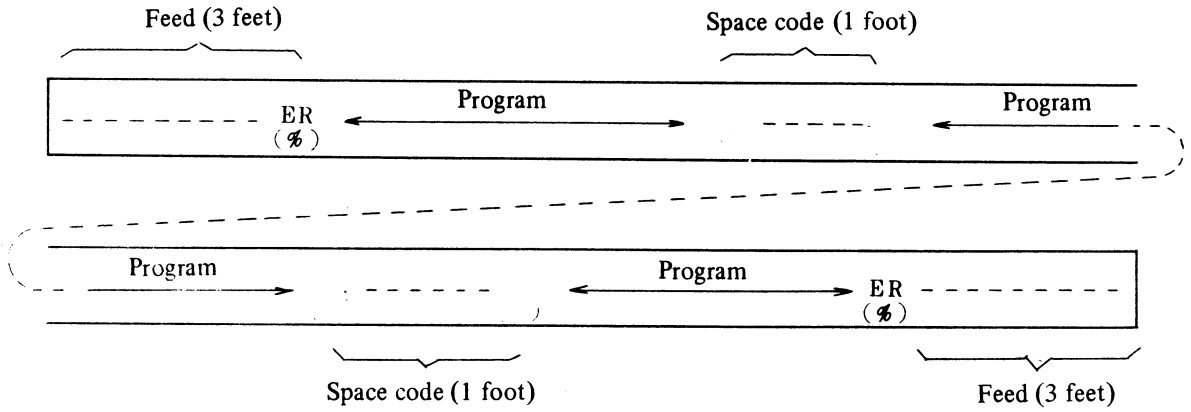
(Note 3) *When the feed of 3 feet is stopped during punch operation, push* CAN .

(Note 4) *When the* RESET *key is pushed, punch operation will stop.*

10.19 Punch of all Programs

All programs in memory can be punched.

(1) Turn on the tape punch unit.
(2) Set the switch of punch panel to MODE 3 and connect the signal cable of tape punch unit.
(3) Set "1" to the setting parameter "I/O".
(4) Set the following parameters according to the tape punch unit used.
Parameter No. 14 "ASR33" and "STP2"
Parameter No. 69 "BRATE 1"
(5) Set the punch code by the setting parameter "ISO".
(6) Set the MODE SELECT switch to EDIT mode.
(7) Select the PRGRM and light the PRGRM LED.
(8) Key-in the address. O.
(9) Key-in [–] [9] [9] [9] [9] [START] .



(Note) The sequence of the program to be punched may not be in sequence.

10.20 Sequence Number Search

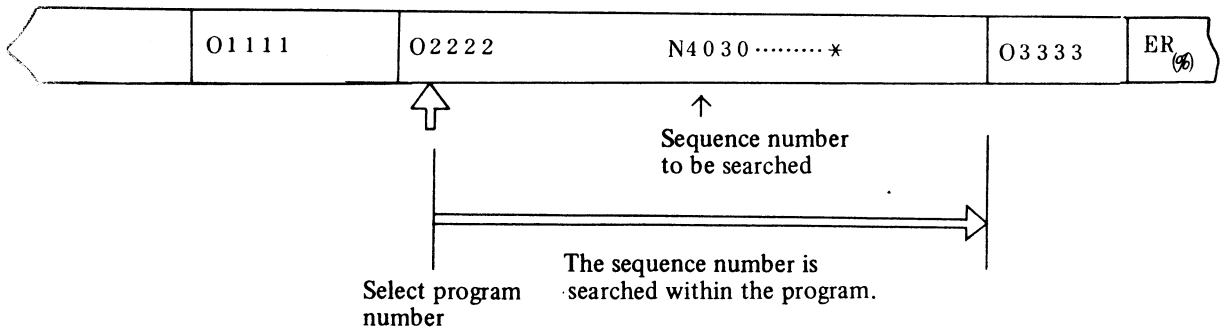
By this function, the sequence number placed in the program is searched so as to start or restart the program from the searched block.

The skipped blocks by this function do not influence the NC. That is, M, S, and T code and G code before the searched block do not change the coordinate value, modal data, etc.

When starting or restarting the program from the searched block after sequence number search, confirm the machine status and set the M, S, and T codes and absolute zero point by MDI operation if necessary. In normal operation, the block to be searched should be at the brakpoint in the process.

If it is desired to search and start a block during process, confirm the states on the machine and the NC, and command the M, S, T or G code and absolute zero point.

Sequence number search in memory operation

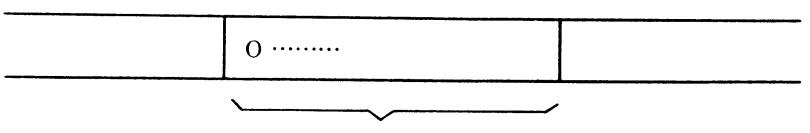


When the sequence number in memory is searched.

(a) Set MODE SELECT switch to AUTO mode.

(b) Select PRGRM and light the PRGRM LED.

(c) Select the program number including sequence number to be searched, by the program number search.

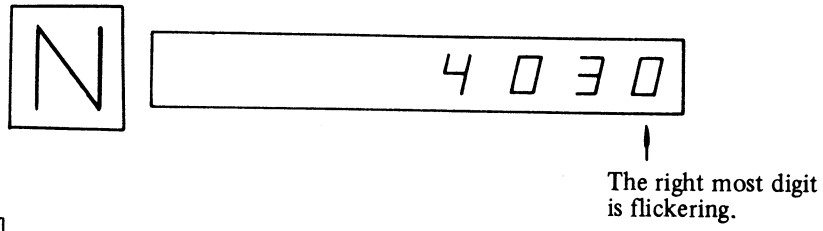


When the desired sequence number exists, follow the next procedure.

If not, the program number which includes the desired number should be searched.

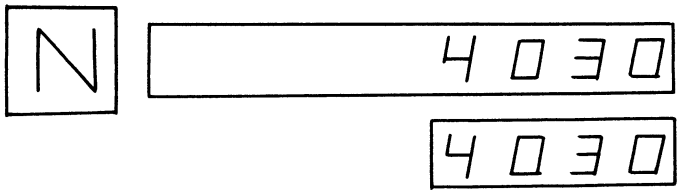
(d) Push the address N.

(e) Input the sequence number to be searched.



(f) Push FWD.

(g) The sequence number search is finished, the searched sequence number is displayed in the number indicator.

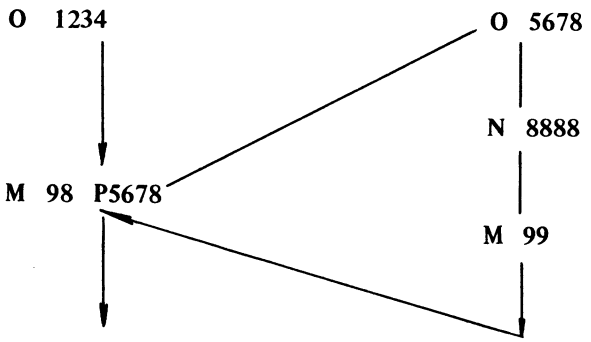


(Note 1) The coordinate values and modal data are not altered. When the sequence number search is finished, these data should be set by MDI if necessary.

(Note 2) The following functions are checked during sequence number search.

- *Optional Block Skip.*
- *P/S alarm (003 ~ 010)*

(Note 3) The subprogram call by M98 command is not executed, during sequence number search. Therefore, when the sequence number search is performed in AUTO mode and the sequence number in the subprogram that is called by the main program is searched, an alarm (alarm number 060) is generated.



In the above example, if the sequence number N8888 is searched, an alarm is generated.

10.21 Collation of Programs

If the program registration operation is performed with the lock key locked, the collation of the program stored in memory and the program on tape is performed.

(1) Set MODE SELECT switch to EDIT mode.

(2) Lock the lock key ( ).

(3) Set the switch of punch panel to MODE 3 and connect the signal cable of portable tape reader.

(4) Set "1" to the setting parameter "I/O".

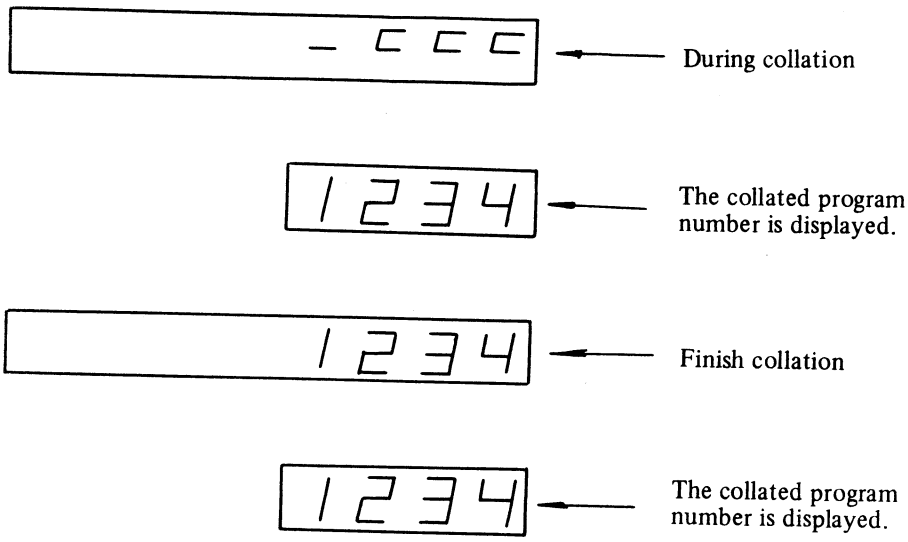
(5) Set the following parameters.

Parameter No. 14	"ASR33" = 0, "STP2" = 1
Parameter No. 69	"BRATE 1" = 4800

(6) Set a program tape to the portable tape reader.

(7) Select the PRGRM and light the PRGRM LED.

(8) Push INPUT key.



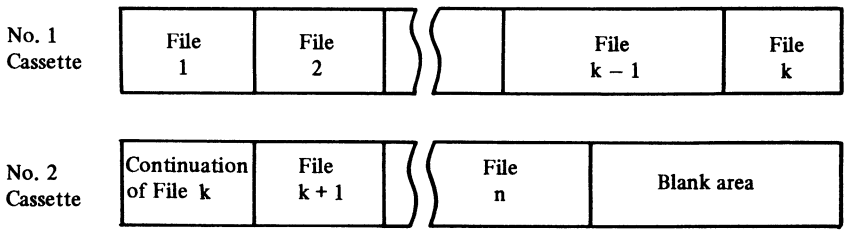
(9) If there are many programs on the tape, the collation is performed until ER (%).

(Note 1) If there is disagreement between programs in collation, P/S alarm (alarm number 79) is generated and the collation is interrupted.

(Note 2) The above operation is performed with the lock key released, the program on the tape is registered and the collation is not performed.

10.22 Program Input from and Output to Memory Cassette

The unit of program or porgrams transferred between the NC and cassette by a single operation, is called a "file." One file is composed of one or all programs in the NC memory.



As seen in the above figure, more than one file can be written in one cassette, and one file can span two cassettes.

10.22.1 File search

Use the following procedure in order to input a desired file.

(1) Set the MODE SELECT switch to EDIT or AUTO mode.
(2) With the adapter power switch off, connect the connector cable to the connector of punch panel.
(3) Set the adapter power switch to ON.
(4) Set the switch of punch panel to MODE 3.
(5) Set "0" to the setting parameter "I/O".
(6) Insert the cassette into the adapter.
(7) Select the PRGRM and light the PRGRM LED.
(8) Key-in the address N.
(9) Key-in the file number to be searched.
(10) Push INPUT key.
(11) Eject the cassette from the adapter.
(12) Set the adapter power switch to OFF.
(13) Disconnect the connector cable from the connector of punch panel.

Thus file search is performed according to the input file number as follows.

(a) N0
The cassette head comes out.

(b) N1 to N9999
File 1 to 9999 is searched out.

(c) N–9999
The file following the one accessed immediately before is searched out.

(d) N – 9998
After – 9998 is specified, N – 9999 is automatically inserted each time a file is input or output. Specifying (a), (b), or (c), or resetting disables this function.

(Note 1) If files exist in the cassette, N0 and N1 provide the same results. However, if no files exist, N1 causes an alarm because of the absence of file 1; and N0 causes the cassette head to come out with no alarm, whether files exist or not. N0 is used when outputting files to a new cassette, or canceling all existing files to output files from the head of the cassette.

(Note 2) If an alarm is caused (e.g., the file is not found) during file search, the NC does not enter alarm status immediately, and issues P/S alarm 86 during file input/output operation.

(Note 3) Sequential file search by specifying N1 to N9999, and search by specifying first N1 to N9999 and then N-9999 for the subsequent files, provide the same results, although requiring different time periods.
(The latter case provides faster search.)

(Note 4) Don't turn the adapter power switch on/off, with the switch of punch panel set to MODE 1. Turn it on/off, with MODE 2 or MODE 3.

10.22.2 Program output to memory cassette

Use the same procedures as for punching programs described in Sections 10.18 and 10.19 in order to output programs to the cassette.

(1) For output of one program

O → Program Number → START

(2) For output of all programs

O → –9999 → START

Thus, following the existing files new files are added. When writting files in a new cassette, or canceling all existing files and writting files from the head, use N0 and then the above procedures.

(Note 1) When P/S alarm 86 occurs during program output operation, the cassette has been restored to the status before output.

(Note 2) After N1 to N9999 is specified, the new file is output to the specified n-th number. Files 1 to (n – 1) are in effect, but the old n-th and subsequent files are canceled. When an alarm occurs during output operation, only files 1 to (n – 1) are restored.

(Note 3) The cassette is provided with a write protect slide. Check that the red lamp is alight when the cassette with this slide set upward is inserted into the adapter, before output operation.

(Note 4) To input an output file again or to collate it with the NC memory, it is recommended that the pertinent file number be noted correctly on the cassette memory label immediately after output.

(Note 5) For efficient use of the cassette memory, set NFED of parameter No. 5 or 14 to 1 so that feed holes are not output with programs.

10.22.3 Program input from memory cassette

After file search in EDIT or AUTO mode, use the same procedure as for paper tape input of Sections 10.13 and 10.14, in order to input programs from the cassette to the NC memory.

(1) Search out the file.

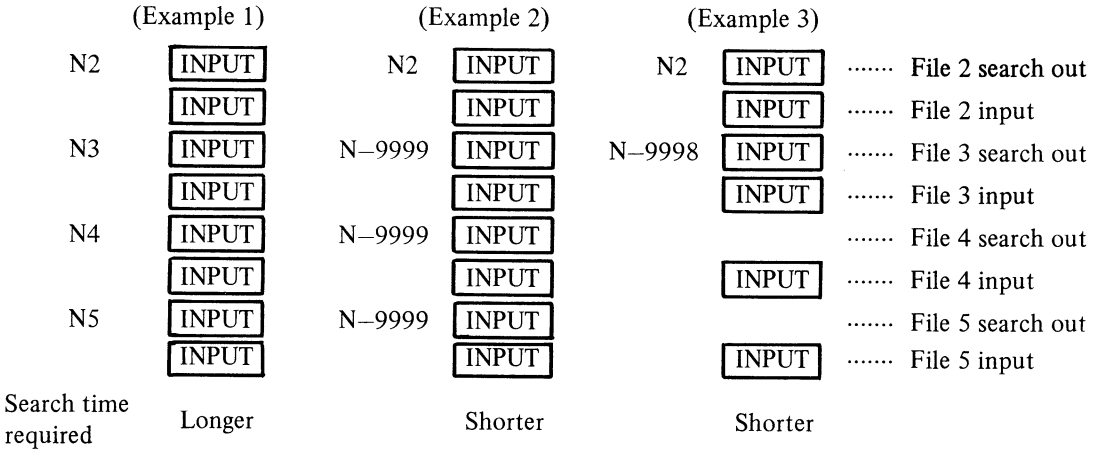
N1 to N9999, N – 9999, or N – 9998 INPUT

(2) Press key INPUT again. (Programs are input.)

(To change the program number, O → Program number → INPUT)

(Note) The above input operation with the lock key () closed, collates the programs in the NC memory and cassette (see Section 10.2).

Examples of registering files 2 to 5 into the memory:



10.22.4 Deletion of file in memory cassette

(1) Set the MODE SELECT switch to EDIT mode.
(2) With the adapter power switch off, connect the connector cable to the connector of punch panel.
(3) Set the adapter power switch to ON.
(4) Set the siwtch of punch panel to MODE 3.
(5) Set "0" to the setting parameter "I/O".
(6) Insert the cassette into the adapter.
(7) Select the PRGRM and light the PRGRN LED.
(8) Open the lock key ( ).
(9) Key-in the address N.
(10) Key-in the file number of the file to be deleted, such as 1 to 9999.
(11) Push START key.
(12) Eject the cassette from the adapter.
(13) Set the adapter switch to OFF.
(14) Disconnect the connector cable from the connector of punch panel.

Thus the file of the keyed-in number is deleted.

(Note 1) After the k-th file is deleted, 1 is subtracted from the old file number (k + 1) to n, resulting in files k to (n – 1).

Before deletion	*After deletion*
1 to (k – 1)	*1 to (k – 1)*
k	*Deleted*
(k + 1) to n	*k to (n – 1)*

(Note 2) As with output, deletion requires the write protect slide to be set upward. (RECORD side)

(Note 3) Don't turn the adapter power switch on/off, with the switch of punch panel set to MODE 1. Turn it on/off with MODE 2 or MODE 3.

10.22.5 Cassette change request

If one file spans two cassettes, after the input/output operation for the first cassette, the red and green lamps of the cassette adapter turn on and off alternately, indicating cassette change request. Remove the first cassette and insert the second. The input/output operation continues.

This change request is output when the second cassette is required during any of the following operations:

① File search
② Program input
③ Program output
④ File deletion

(Note 1) Cassette change is handled by the adapter. Therefore, the NC need process nothing (the NC only interrupts input/output processing).

(Note 2) If a reset operation is conducted for the NC during cassette change request, the NC is not reset immediately but after the change. Therefore, to delete change request, only the cassette change is required.

10.22.6 Cassette adapter lamp status

The cassette adapter is provided with two lamps, red and green, indicating operation status.

(1) When the red lamp lights.
Ready for writing . . . . . This lamp lights if the cassette is inserted with the write protect slide set upward. (RECORD side)

(2) When the green lamp lights.
Ready for reading . . . . . This lamp lights if the cassette is inserted.

(3) When the red lamp flickers.
Writing

(4) When the green lamp flickers.
① Reading
② File searching

(5) When the red and green lamps turn on and off alternately.
① Cassette yet to be inserted
② Cassette change request

(6) When the red and green lamps turn on and off simultaneously.
File deleting

(Note) Whenever an error is detected in the cassette adapter, the NC issues P/S alarm 86. Therefore, trace the cause from the cassette status and operation, and remove it.

11. PROGRAM EDITING OF NC LANGUAGE

The contents of a program stored in memory can be altered.

(1) Set MODE SELECT switch to EDIT mode.

(2) Select the PRGRM.

(3) Select the program.

When the program to be edited is not selected, the program number should be searched. (When selected, it is not necessary.)

(4) Search for the word to be altered.

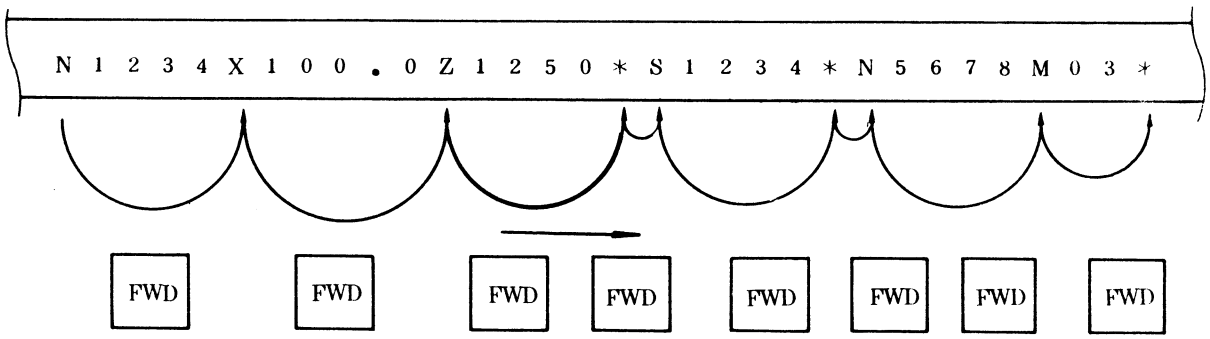
(a) Method of SCAN.

(b) Method for word search.

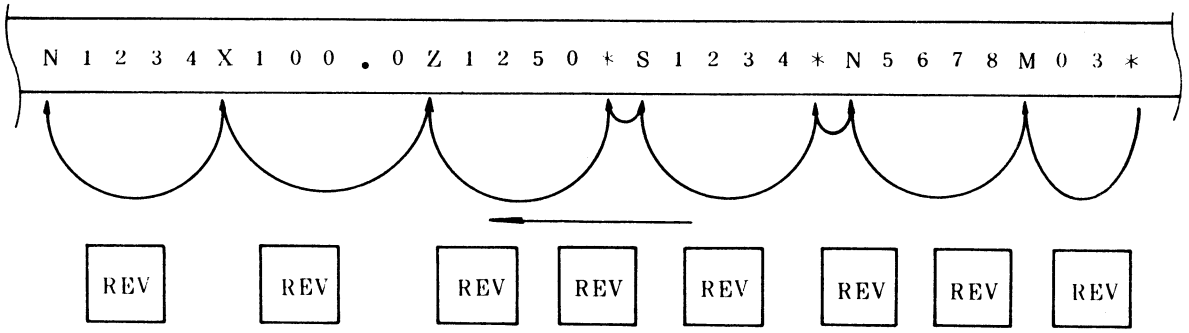
(5) The editing operation, alteration, insertion or deletion of word can be performed as follows.

11.1 Method of SCAN

(1) When [FWD] is pushed.



(2) When [REV] is pushed.



Display

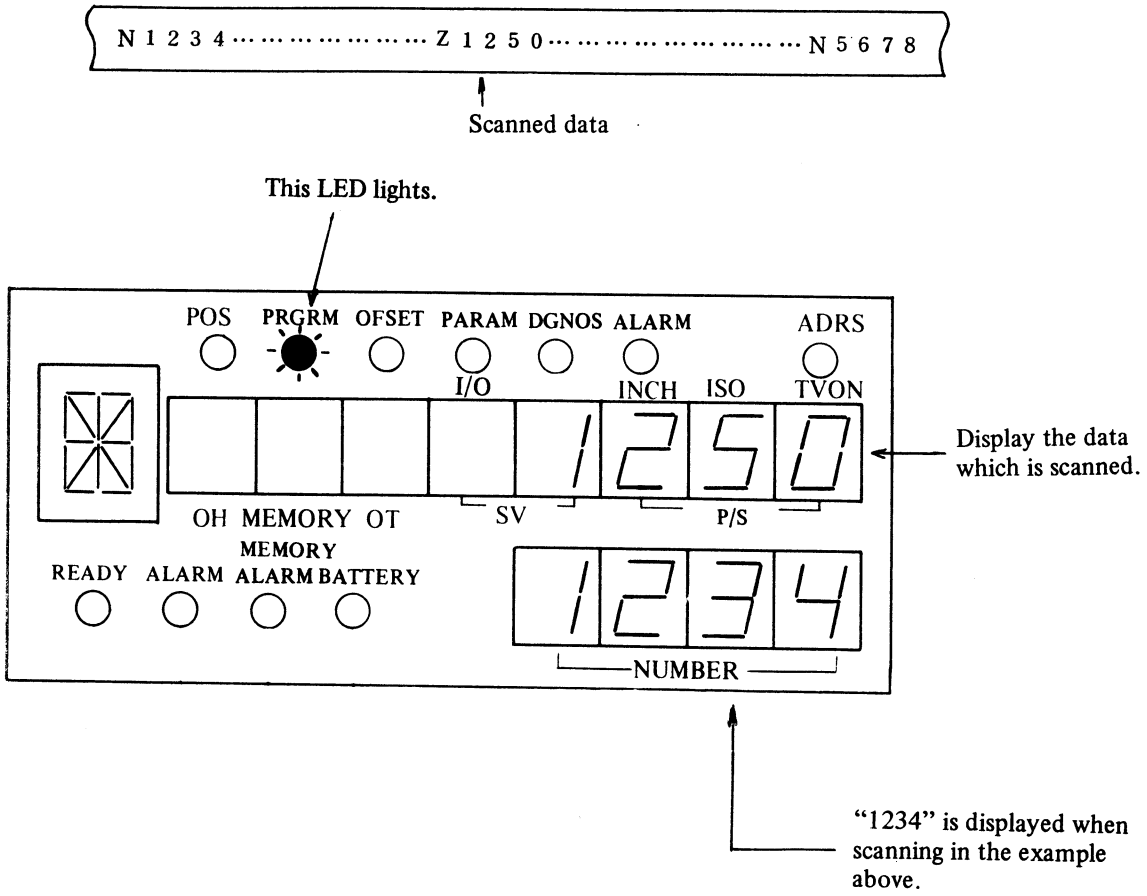
The selected address and the data are displayed in the data indicator.

However, EOB and ER are displayed as follows:

EOB . . . . . * is displayed.

ER . . . . . □/□ is displayed.

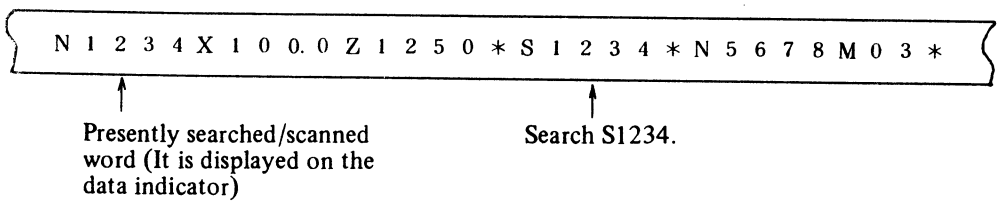
(Example)



(3) If keeping pushing the FWD or REV , the search is performed continuously.

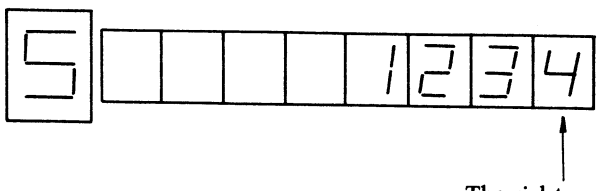
11.2 Method for Word Search

Specified word can be searched from the present position in the normal direction.



(1) Key-in the address S. → Direction of search

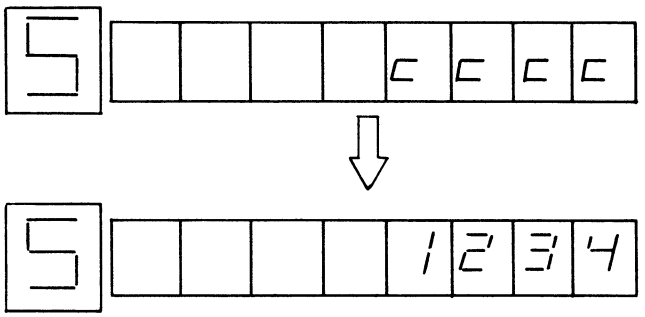
(2) Key-in 1 U 2 W 3 R 4 X



(Note 1) The word 'S1234' can not be searched only by keying in 'S123'.

(Note 2) When searching S009, don't omit the leading zero. That is, don't input 'S9' but 'S009'.

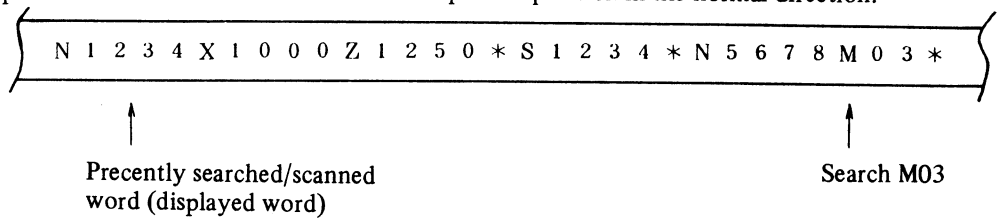
(3) Push FWD , and the search begins to start.



If REV is pushed instead of FWD key, the word search is performed in the reverse direction.

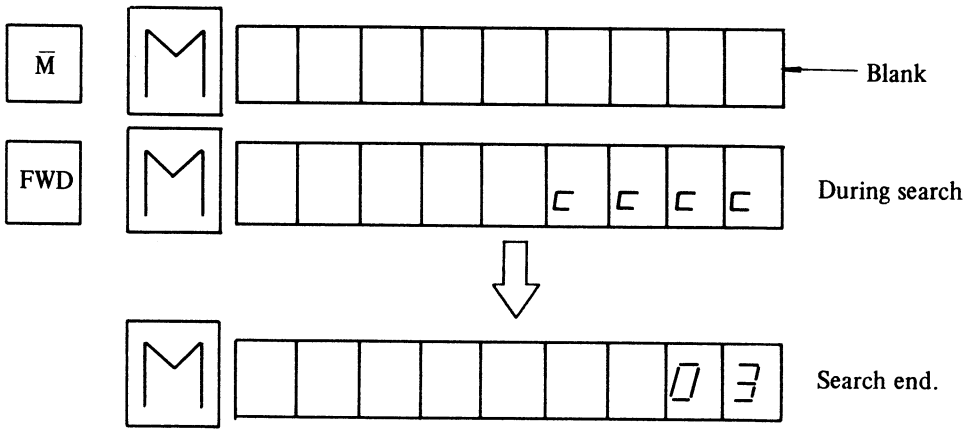
11.3 Method of Address Search

Specified address can be searched from the present position in the normal direction.



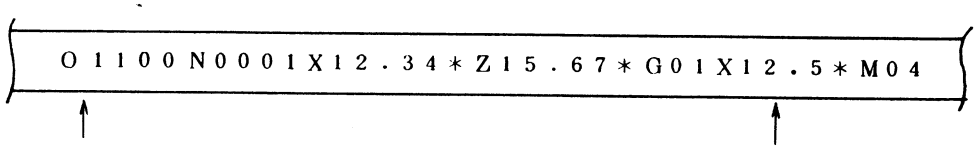
(1) Key-in the address M.

(2) Push FWD .



If REV is pushed instead of FWD key, the address search is performed in the reverse direction.

11.4 Method for Return to the Start of a Program



(1) Method 1.

(a) Push RESET . (In EDIT mode and select PRGRM).

The program number is displayed on the data indicator after return to the start of the program is finished.

(2) Method 2

Search for a program number.

(3) Method 3

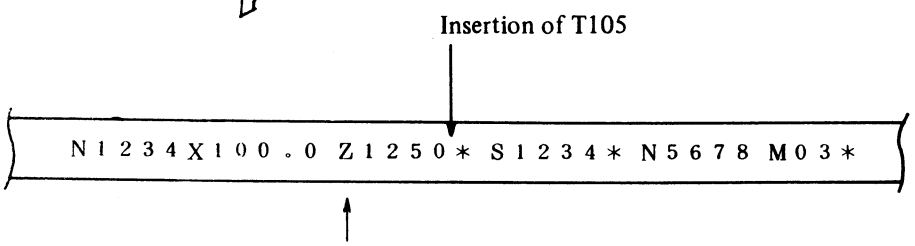
(a) Set MODE SELECT switch to AUTO or EDIT mode.

(b) Select the PRGRM.

(c) Key-in the address O.

(d) Push the [REV] .

11.5 Insertion of Word ( )



(1) Search for the word that is before the placing in the program in which the new word is to be inserted.

(a) Method by SCAN

Refer to item 11.1.

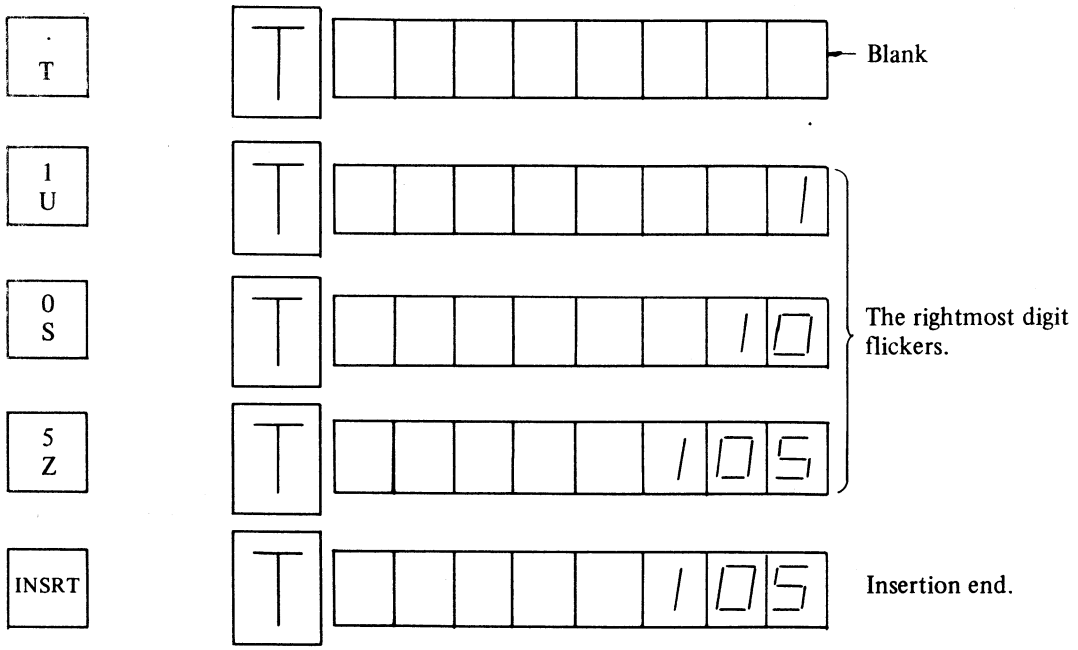
(b) Method by word search.

Refer to item 11.2.

(2) Key-in the address to be searched.

(3) Input the data.

(4) Push [INSRT]



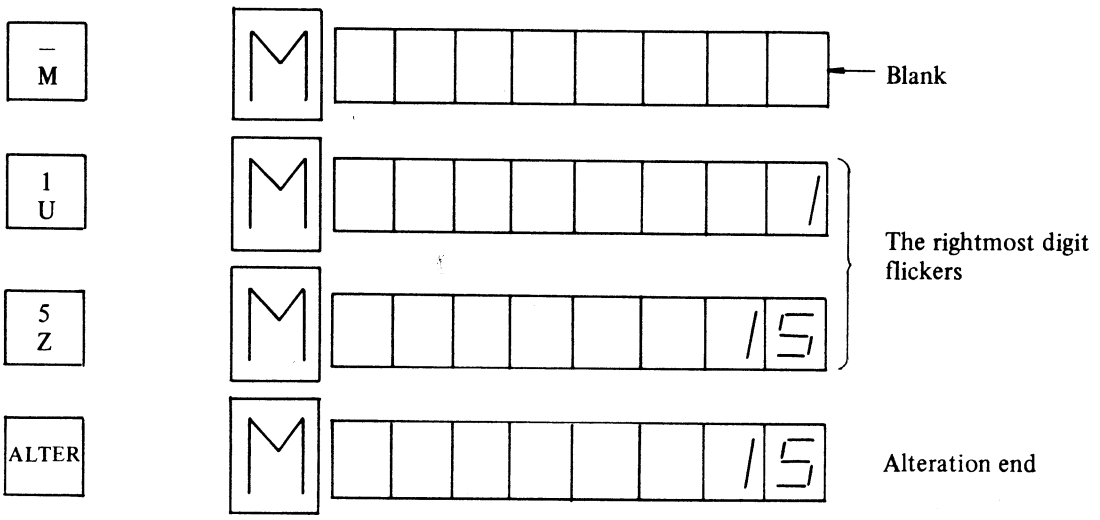
(5) Confirm the placement of the word and the words before and after by using [REV] and [FWD].

```
N1234X100.0Z1250T105*S1234*N5678M03*
```

11.6 Alteration of Word ( )

```
N1234X100.0Z1250T105*S1234*
                  ↑
          Alter to M15.
```

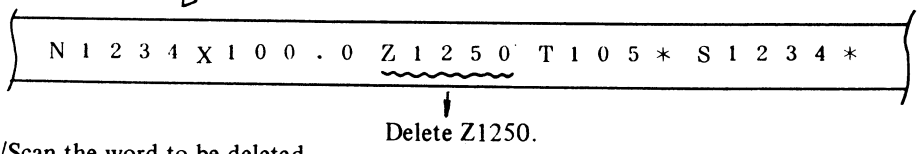
(1) Search/Scan the word to be altered.
(2) Key-in the address to be altered.
(3) Input the data.
(4) Push [ALTER].



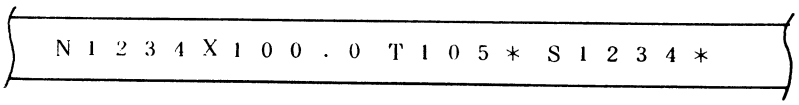
(5) Confirm the altered word and the words before and after, by using [FWD] and [REV].

```
N1234X100.0Z1250M15*S1234*
```

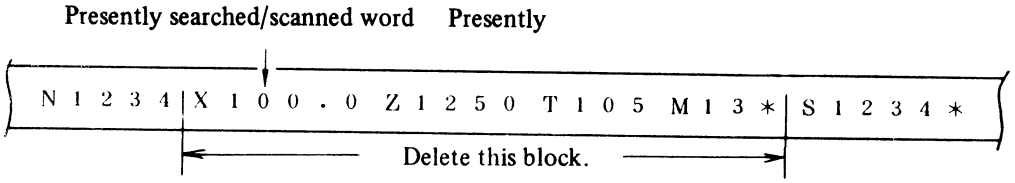
11.7 Deletion of Word ( )



(1) Search/Scan the word to be deleted.
(2) Push DELET .
After the DELETE is pushed, the next word is displayed in the data indicator.
(3) Confirm the words before and after by FWD and REV .



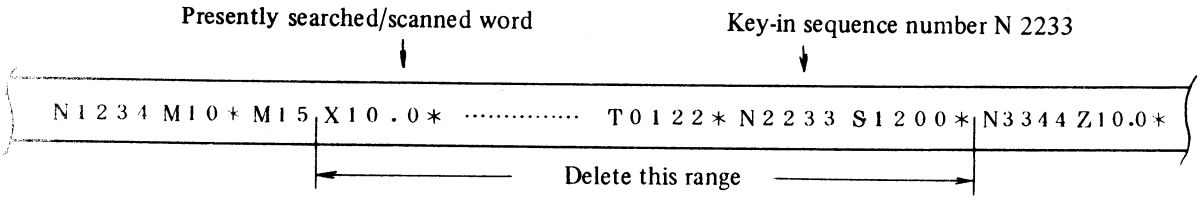
11.8 Deletion until EOB Code ( )



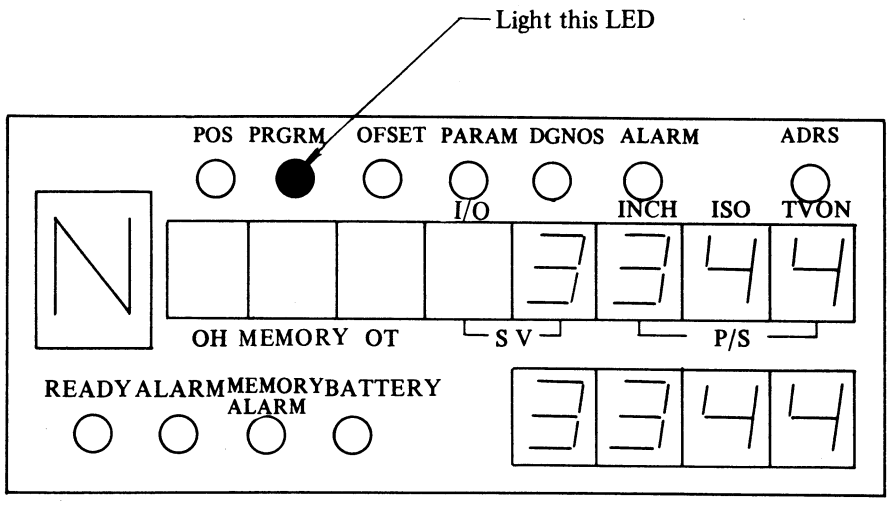
If the / EOB , DELET is pushed in sequence, all words until the next EOB are deleted and the next word is displayed.

11.9 Deletion of Several Blocks ( )

From the present display word to the block of a specified sequence number can be deleted.



(1) Key-in the address N.
(2) Input the sequence number "2233" in the example above.
(3) Push DELETE .
When DELETE is pushed, everything from the present display word to the block of N2233 is deleted and the next word is displayed by the data indicator.



11.10 Display of Memory Capacity

The program memory capacity is displayed by the following procedure.

(1) Set the MODE SELECT switch to EDIT mode.
(2) Select the PRGRM and light PRGRM LED.
(3) Key-in the address P.
(4) Key-in 1, 2, 3 or 4. The displayed contents vary according to the number shown in table below.
(5) Push INPUT key.

The followings are displayed according to the number.

Address	Number	Displayed contents
P	1	The number of programs currently stored in the program memory (including the number of subprograms)
	2	The number of programs to be stored moreover (Remained capacity)
	3	The program memory capacity currently used. (in number of characters)
	4	The program memory capacity to be stored moreover. (in number of characters)

V OPERATION IN MACHINE OPERATOR'S PANEL

1. OPERATION OF THE OPERATOR'S PANEL

1.1 Operator's Panel

The operator's panel varies in functioning and switch arrangement between the different machine tools.

Operations of a typical operator's panel is explained in Fig. 1.1.1. However, for details, refer to the manual issued by the machine tool builder.

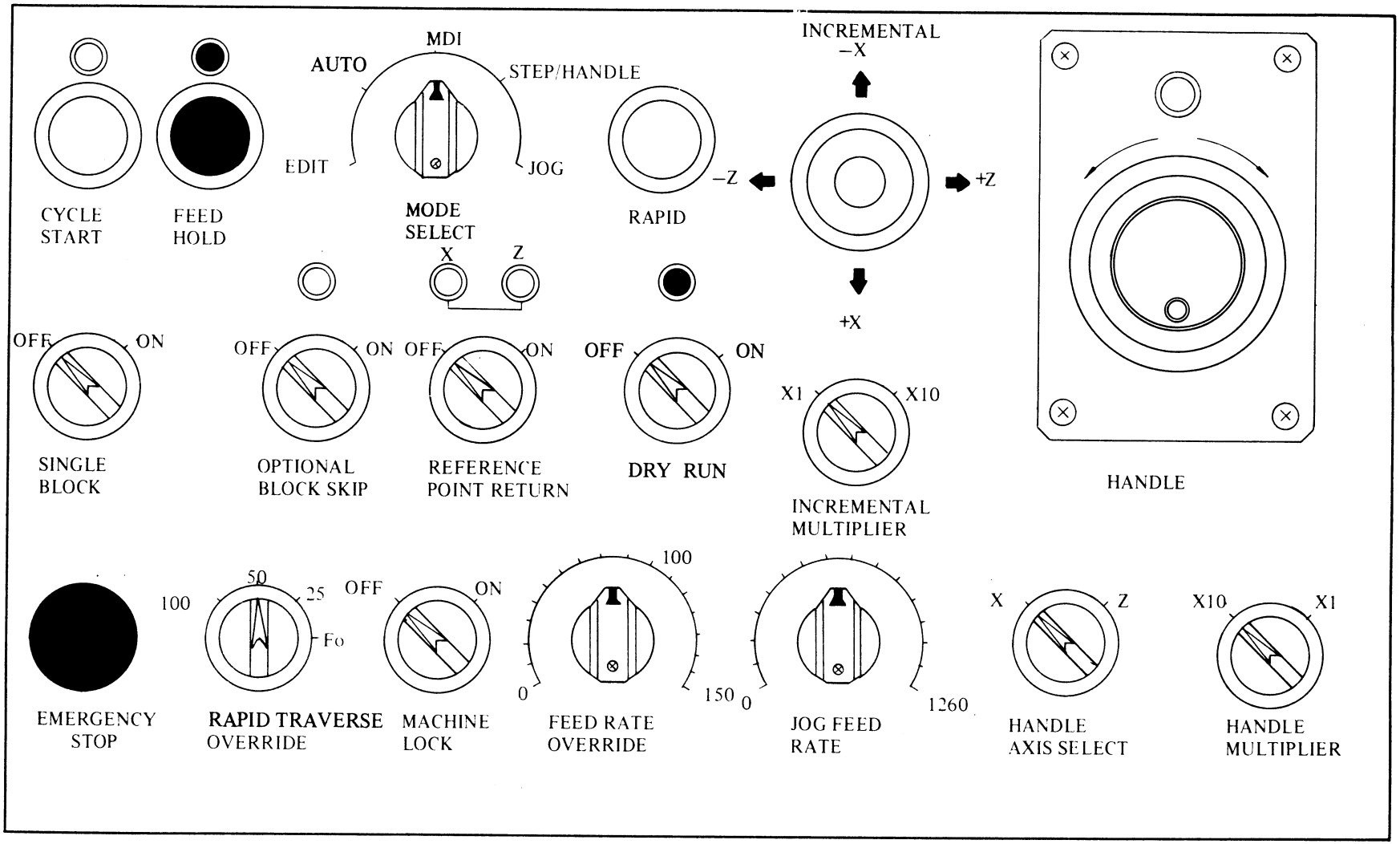
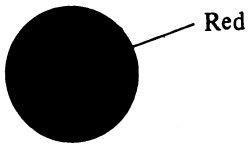


Fig. 1.1.1 Example of Operator's Panel.

1.2 Emergency Stop

This button is pushed to make a stop in case of emergency.



EMERGENCY STOP

If this button is pushed, the feed is stopped immediately, and it is locked.

The method of release varies with the machine tool builder.

Usually, this button is released by turning it clockwise with keeping pressing.

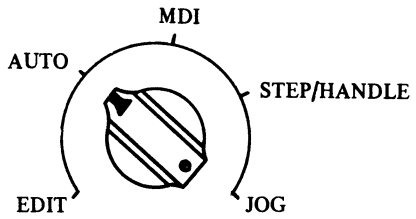
(Note 1) *If this button is pushed, the current to the motors is interrupted.*

(Note 2) *The control unit assumes a reset state.*

(Note 3) *Remove fault causes before the button is released.*

(Note 4) *After the button is released, reference point return by manual operation or G28 should be commanded.*

1.3 Mode Select



This switch is used to specify an operational mode.

Mode	Function
EDIT	(i) Registration of program to memory. (ii) Modification, addition and deletion of program. (iii) Punch-out of program in memory. Other operations for editing can be performed.
AUTO	(i) The program stored in memory can be executed. (ii) The sequence number and program number search of the program in memory can be executed.
MDI	(i) Operation by commands on the MDI & DPL panel can be executed.
STEP/HANDLE	(i) HANDLE feed or INCREMENTAL feed can be executed.
JOG	(i) JOG feed can be executed.

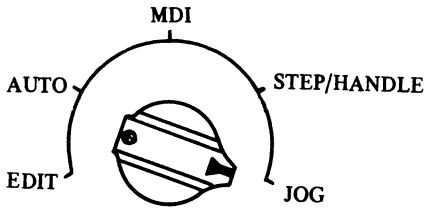
1.4 Operation Relative to Manual Operations

The following manual operations are possible, using the switches on the operator's panel.

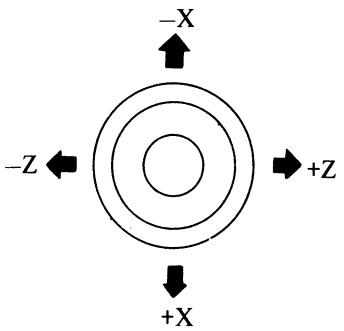
1.4.1 JOG feed

It is possible to move the machine tool continuously.

(1) Set MODE SELECT switch to JOG position.



(2) Select the movable axis.

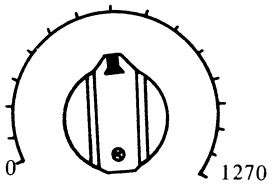


The selected axis is moved in the selected direction while this switch is kept turned on.

(Note 1) In manual operation only one axis can be controlled at a time.

(Note 2) If the MODE SELECT switch is changed to JOG mode at turning power on, the machine does not move even if the axis has already been selected. In this case, select the axis again.

(3) Select JOG feed rate.



JOG
FEED RATE

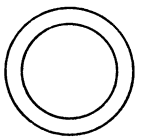
(inch/min)

Position on Rotary switch	JOG feedrate			
	Metric ball screw		Inch ball screw	
	mm/min	inch/min	inch/min	mm/min
0	0	0	0	0
1	2.0	0.079	0.08	2.03
2	3.2	0.126	0.12	3.05
3	5.0	0.197	0.2	5.08
4	7.9	0.311	0.3	7.62
5	12.6	0.496	0.5	12.7
6	20	0.787	0.8	20.3
7	32	1.26	1.2	30.5
8	50	1.97	2.0	50.8
9	79	3.11	3.0	76.2
10	126	4.96	5.0	127
11	200	7.87	8.0	203
12	320	12.6	12	305
13	500	19.7	20	508
14	790	31.1	30	762
15	1260	49.6	50	1270

(Note 3) The feedrate error (about ±3%) affects on the feedrate in the table above.

(4) Rapid traverse

It is possible to move the axis in the selected direction while this button is kept pushed.



RAPID
(RAPID TRAVERSE button)

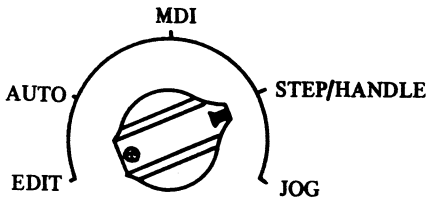
The selected axis is moved at the rapid traverse during pushing this button.

(Note 4) Time constant and method of automatic acceleration/deceleration, and the feedrate for manual rapid traverse are the same as G00 in G codes.

(Note 5) After turning power on or releasing the emergency stop state, if reference point return is not performed, the machine moves not at the rapid traverse but the jog feed.

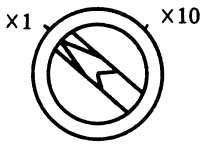
1.4.2 INCREMENTAL feed

(1) Set MODE SELECT switch to STEP mode.



INCREMENTAL feed is available for the machine without manual pulse generator.

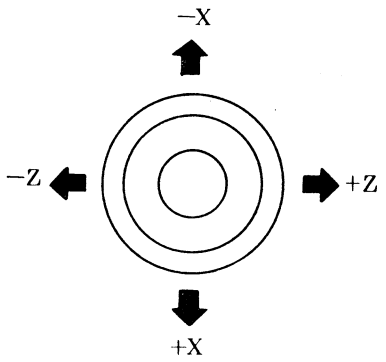
(2) Select the movement amount.



Input system	x 1	x 10
Metric input	0.001 mm	0.01 mm
Inch input	0.0001 inch	0.001 inch

(Note 1) In diameter designation, programmed move amount in X axis is a decrement of diameter amount.

(3) Select the movable axis.



The selected axis is moved selected distance in the selected direction while this switch is turned on.

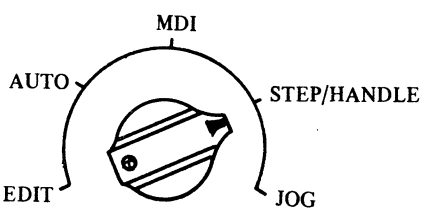
(Note 2) Incremental feed rate is same as JOG feed rate.

(Note 3) Rapid traverse button can be used. In this case, the rapid traverse override is available.

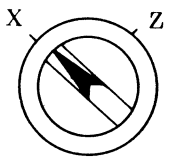
1.4.3 HANDLE feed

By the use of a manual pulse generator, fine adjustment feed is available for the machine tool.

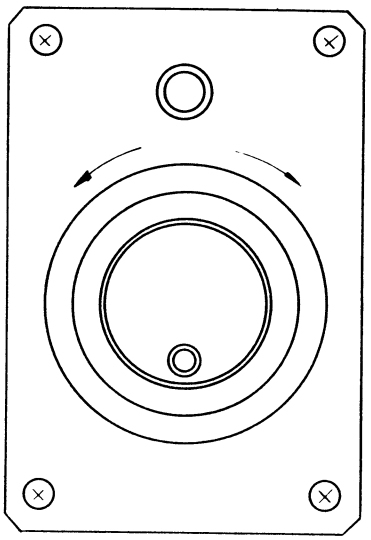
(1) Set MODE SELECT switch to HANDLE mode.



(2) Select the movable axis.



(3) Rotate the handle of the manual pulse generator.



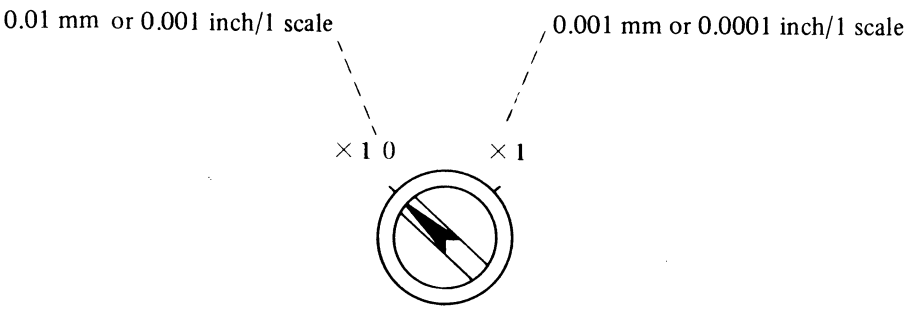
clockwise rotation . . . . . + direction
counterclockwise rotation . . . . . – direction
(The direction varies with machine tool builder.)

(4) Movement amount

Input system	Movement amount per scale
Metric input	0.001 mm
Inch input	0.0001 inch

(Note) In diameter programming, actual movement amount is 0.0005 mm or 0.00005 inch.

"X10" switch is provided on the operator's panel. Movement amount is multiplied ten times by "X10" switch.



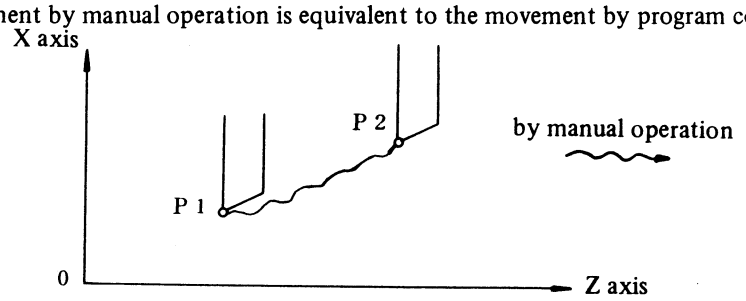
HANDLE MULTIPLIER

(Note 1) *If the handle is rotated in excess of 5 rotation/sec, there arises a difference between handle rotation amount and machine movement distance.*

(Note 2) *When optional manual pulse generator is equipped, Incremental feed function is not used.*

1.4.4 Movement by manual operation

Movement by manual operation is equivalent to the movement by program command.

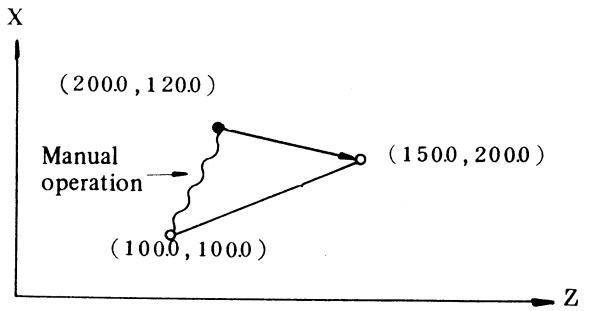


The coordinate value varies the value of manual operation.

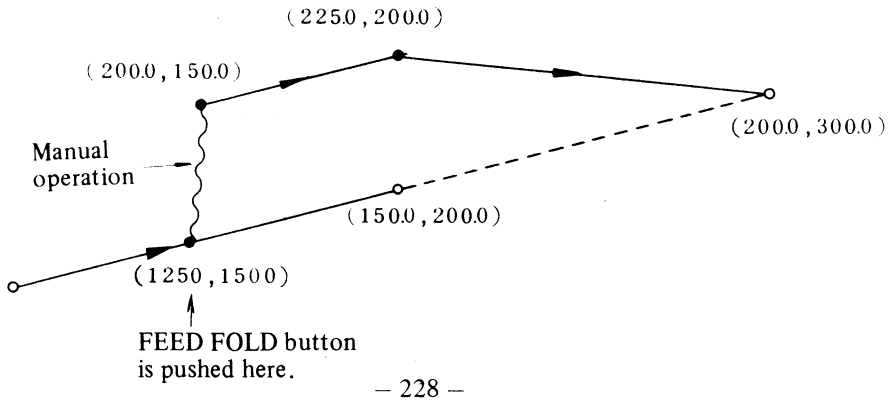
Example: Suppose the next tape commands:

```
.
.
.
G01X100.0Z100.0F010 *    ①
    X150.0Z200.0    *    ②
    X200.0Z300.0    *    ③
.
.
.
```

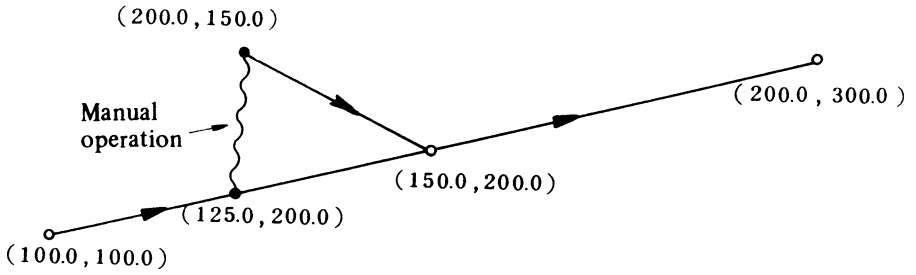
(a) Upon completion of the shift in block ①, block ② is executed after a manual operation process (+100.0 in the X-axis direction, +20.0 in the Z-axis direction).



(b) The FEED HOLD button is pushed during the execution of block ②, and the CYCLE START button is again pushed after a manual operation process (X + 75.0).

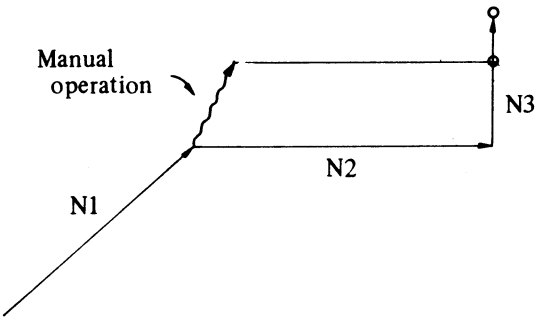


(c) The FEED HOLD button is pushed during the execution of block ②, and a resetting action is taken, after a manual operation process (X + 75.0), with the RESET button to read block 2 again.



(d) When the command following the manual operation is one axis command, only the commanded axis returns to the programmed position of the axis.

```
N1  G01 X 100. 0 Z100. 0 F50*
N2  Z200. 0*
N3  X150. 0*
```

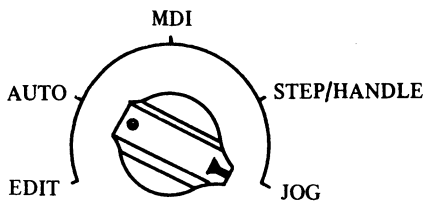


(e) When the command following the manual operation is incremental command, the axis moves the incremental distance from the position after manual operation.

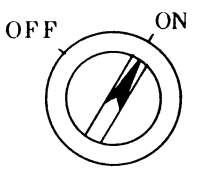
1.5 Manual Reference Point Return

The machine can be returned to the reference point by manual operation.

(1) Set MODE SELECT switch to the JOG mode.

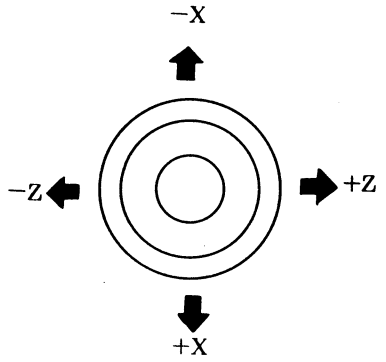


(2) Turn on REFERENCE POINT RETURN switch.



REFERENCE POINT RETURN Switch

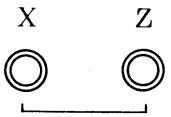
(3) Move each axis to the reference point at JOG feed.



This button should be turned on until the completion of reference point return.

The selected axis is moved to the decelerating point at rapid traverse, and then to the reference point at FL speed. The rapid traverse override is effective at rapid traverse.

(4) The machine stops at the referent point with the REFERENCE POINT RETURN COMPLETION LED lighting.



REFERENCE POINT RETURN COMPLETION LED

(Note 1) After completing reference point return and lighting the completion LED, if the reference point return switch is not turned OFF, the machine cannot be moved.

(Note 2) REFERENCE POINT RETURN COMPLETION LED is extinguished by the following operations.

(1) Moving from the reference point.

(2) Depressing the emergency stop button.

(Note 3) For the distance of return to reference point, refer to the manual issued by the machine tool builder.

(Note 4) If the overtravel is generated by the tool movement at turning power off, the reference point return cannot be performed at turning power on.
In this case, turn on the power again with [Q/P] *and* [CAN] *keys pressed, and perform the reference point return. Because the overtravel check function becomes ineffective until the reference point return is completed.*

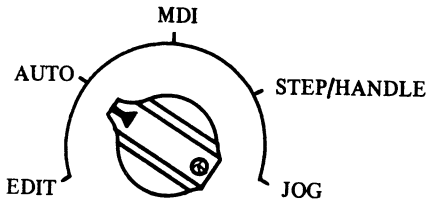
1.6 Operation Relative to Automatic Operation

The machine tool can be moved according to programming by automatic operation.

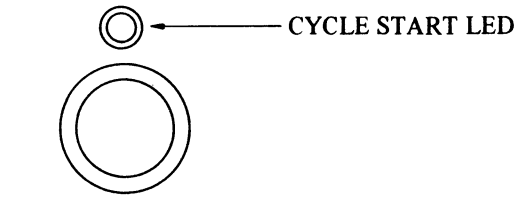
1.6.1 Start of automatic operation.

(i) Select the program.
Refer to item 4.15 Program number search.

(ii) Set the MODE SELECT switch to AUTO mode.



(iii) Push CYCLE START button.



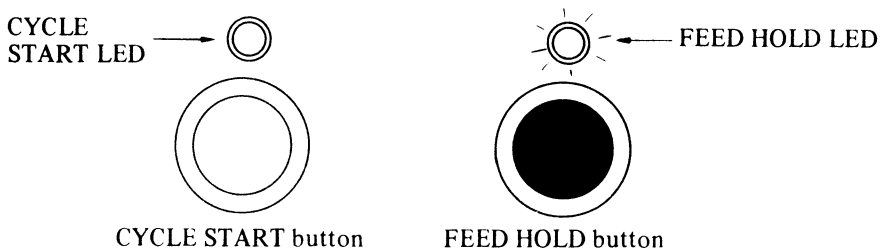
When the CYCLE START button is pushed, automatic operation starts and CYCLE START LED lights.

(Note 1) In the following cases, the CYCLE START button is ineffective.

(a) When FEED HOLD button is pushed.
(b) When EMERGENCY STOP button is pushed.
(c) When RESET signal is turned ON. (For details, contact the machine tool builder).
(d) When MODE SELECT switch is set to a abnormal position.
(e) When a sequence number is being searched.
(f) When an alarm has occurred.
(g) When automatic operation is selected.
(h) When NC doesn't become READY state.

1.6.2 Haulting the automatic operation

(1) Push FEED HOLD button



When FEED HOLD button is pushed, the CYCLE START LED is extinguished and FEED HOLD LED is lit.

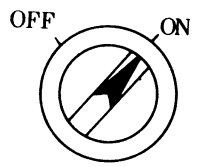
(i) The feed is stopped after deceleration if the machine tool is moving.
(ii) Dwell is not continued even in a feed hold state if the dwell is being executed.
(iii) The machine tool is stopped after the operation of the M, S or T function is executed.

(Note 1) Even if the FEED HOLD button is pushed during thread cutting by G32 or G92, the feed is not stopped at the time that the button is pushed. The feed is stopped after completion of the thread cutting and, if it is pushed at the block not related with thread cutting, the feed is stopped at the time.

1.6.3 Single block

When this switch is on, the control executes only one block of information each time the cycle start button is depressed.

(1) Turn on SINGLE BLOCK switch.



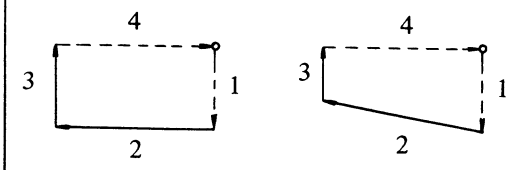
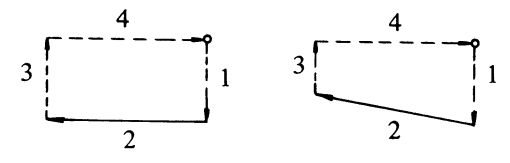
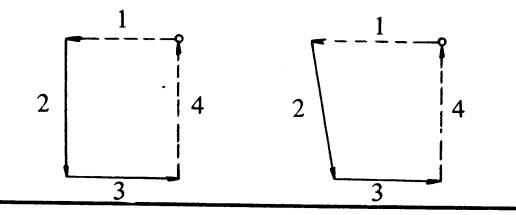
SINGLE BLOCK switch

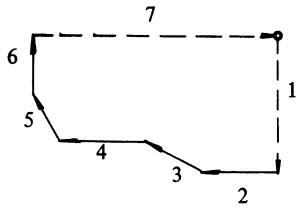
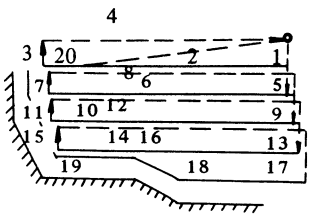
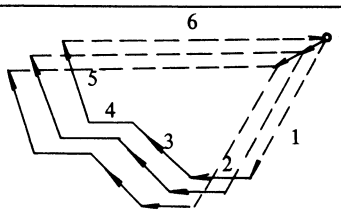
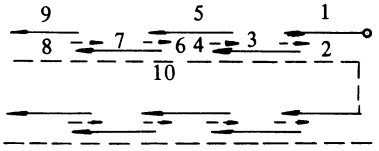
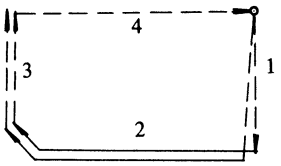
The machine stops after one block execution by pushing CYCLE START button.

(Note 1) If SINGLE BLOCK switch is set to ON during thread cutting by G32 or G92, the feed is not stopped at the position. The feed is stopped after the thread cutting and the execution of the next non thread cutting block.

(Note 2) If SINGLE BLOCK switch is set to ON during canned cycle (G90, G92, G94), the actual executions are as follows:

---→ Rapid traverse
——→ Cutting feed

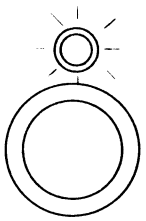
G code	Tool path	Explanation
G90		Tool path 1 to 4 is assumed as one cycle. After 4 is finished a stop is made.
G92		
G94		

G70		Tool path 1 to 7 is assumed as one cycle. After 7 is finished, a stop is made.
G71 G72	 *(Note) The above figure shows the case of G71. The figure of G72 is the same as this.*	Tool path of each of 1 to 4, 5 to 8, 9 to 12, 13 to 16, and 17 to 20 is assumed as one cycle. After each cycle is finished, a stop is made.
G73		Tool path 1 to 6 is assumed as one cycle. After 6 is finished, a stop is made
G74 G75	 *(Note) The above figure shows the case of G74. The figure of G75 is the same as this.*	Tool path 1 to 10 is assumed as one cycle. After 10 is finished, a stop is made.
G76		Tool path 1 to 4 is assumed as one cycle. After 4 is finished, a stop is made.

1.6.4 Restart

(1) Return the MODE SELECT switch to the desired operating mode (AUTO mode or MDI mode).

(2) Push the CYCLE START button.

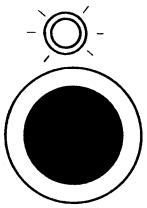


CYCLE START button

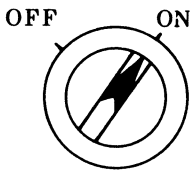
When CYCLE START button is pushed, the FEED HOLD LED is extinguished, and the CYCLE START LED is lit.

1.6.5 Manual operation during automatic operation

(1) Push the FEED HOLD button on the operator's panel, or set the SINGLE BLOCK switch to ON, to make a temporary stop.



FEED HOLD

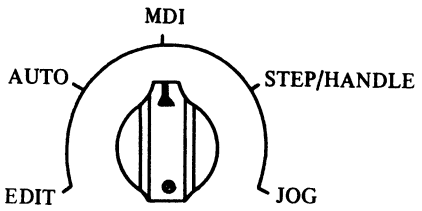


SINGLE BLOCK

(2) Record the coordinate value of the stop position from the position display unit.
(3) Perform the manual operation (see item 1.4.4).
(4) Return the tool to the recorded coordinate value (the start point of the manual operation).
(5) In order to resume the automatic operation, set the MODE SELECT switch to the position it was at prior to the manual operation.
(6) Push the CYCLE START button.

1.6.6 MDI operation during automatic operation

(1) Set the SINGLE BLOCK switch to ON. The machine stops after completion of block.
(2) Set the MODE SELECT switch to MDI mode.



(3) Perform the operation from the Manual Data Input unit.
(4) In order to resume the automatic operation, set the MODE SELECT switch to the position it was at prior to the manual operation.
Push the CYCLE START button on the operator's panel.

(Note 1) As old modal data designated by NC command is remained, new modal data (ex: G code, movement value in canned cycles) must be designated, if necessary.

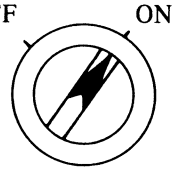
(Note 2) After MDI operation, the modal data (ex: G code, movement value in canned cycles) must be returned to condition it was in before the MDI operation.

(Note 3) MDI operation cannot be performed during feed hold state.

1.6.7 Optional block skip

This is the function which allows the control to skip a block of information in which a slash "/" is punched as the first character in the block.

The block with slash code is effective. ..... OFF ON ..... The block with slash code is ignored.

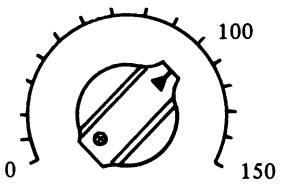


OPTIONAL BLOCK SKIP

(Note 1) The discrimination of this function being effective/ineffective is performed when a block is entered into the buffer from the memory. Therefore, this function is ineffective for the block which has already been entered into the buffer.

1.6.8 Feed rate override

An override in increments of 10% from 10 to 150% can be provided for the feed rate specified by the F function.



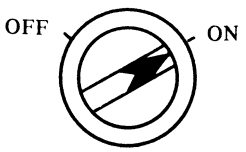
FEED RATE OVERRIDE

(Note 1) A JOG feed rate can also be set with this switch by machine tool builder.
(Note 2) The switch is ineffective during thread cutting and it is clamped at 100%.

1.6.9 Machine lock

With the switch at MACHINE LOCK, move command pulses are suppressed. Consequently, by cycle operation or manual operation, the position indicator is updated as specified, but the machine tool does not move.

This function is used for checking a program.



MACHINE LOCK

(Note 1) When a G27 or G28 command is specified, the machine is not moved to the reference point in machine lock state and the REFERENCE POINT RETURN LED is not lighted.
(Note 2) The M, S and T functions are executed.

1.6.10 Mirror image

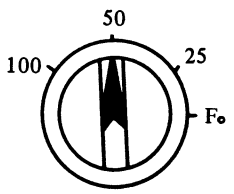
The sign of the X axis move commands can be reversed in cycle operation.

The movement in manual operation and between the intermediate point and reference point in automatic reference point return are not reversed with this function. This function is used when the machine has an opposite tool post.

1.6.11 Rapid traverse override

The rapid traverse override switch of 100%, 50%, 25% and Fo can be provided with the machine operator's panel. When the feed rate is 10m/min. and this switch is set to the position of 50%, the actual feed rate becomes 5 m/min. Fo is the fixed value (feed rate) specified by the each machine tool builder. This function is available in the following cases.

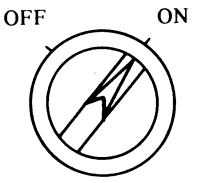
(1) Rapid traverse by G00
(2) Rapid traverse during canned cycle
(3) Rapid traverse in G27 and G28
(4) Manual rapid traverse
(5) Rapid traverse in manual reference point return



RAPID TRAVERSE OVERRIDE

1.6.12 Dry run

If this switch is set to ON, programmed feed rate is ignored and the machine tool is moved at the following speed.



DRY RUN

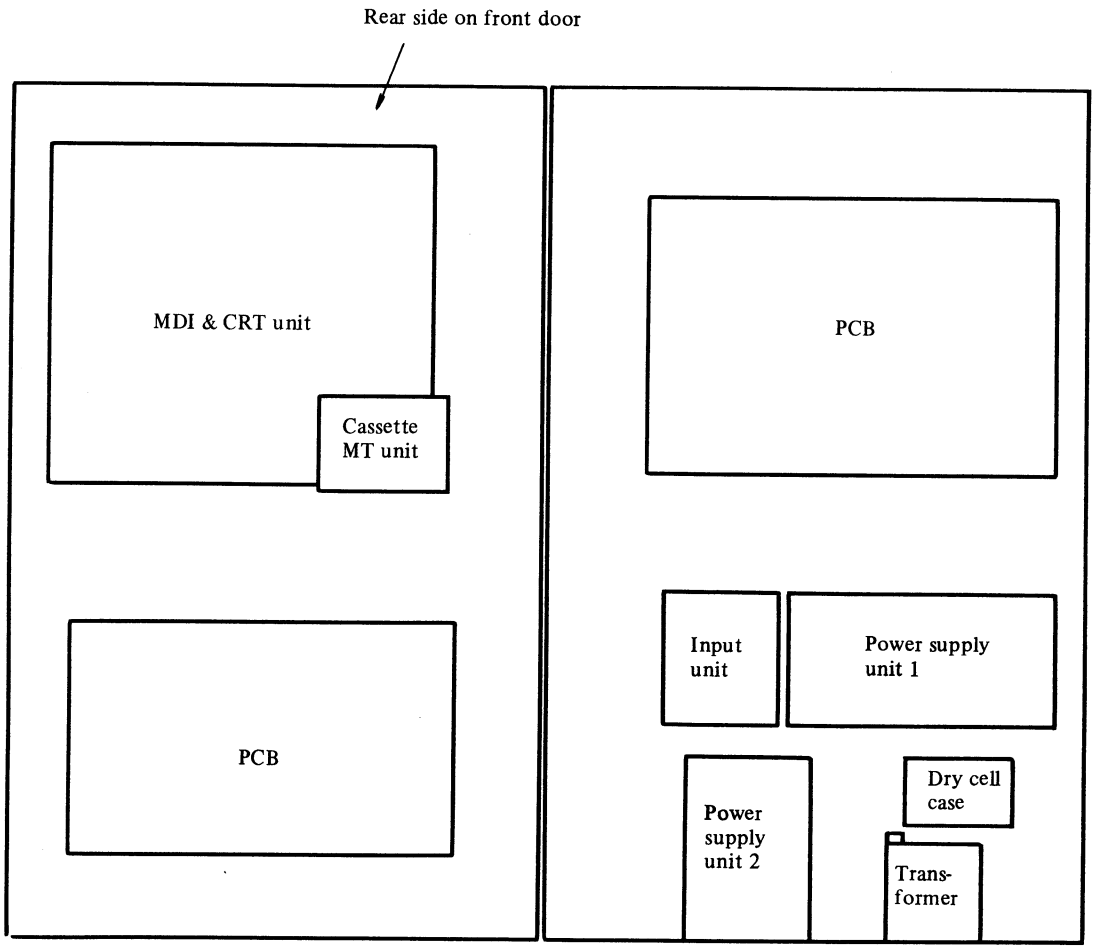
Rapid traverse button ON/OFF	Rapid traverse	Cutting feed
ON	Rapid traverse	Max. JOG feedrate
OFF	JOG feedrate (Note)	JOG feedrate

(Note) Dry run for rapid traverse can be made effective or ineffective by parameter setting (RDRN of parameter No. 04).

VI. MAINTENANCE

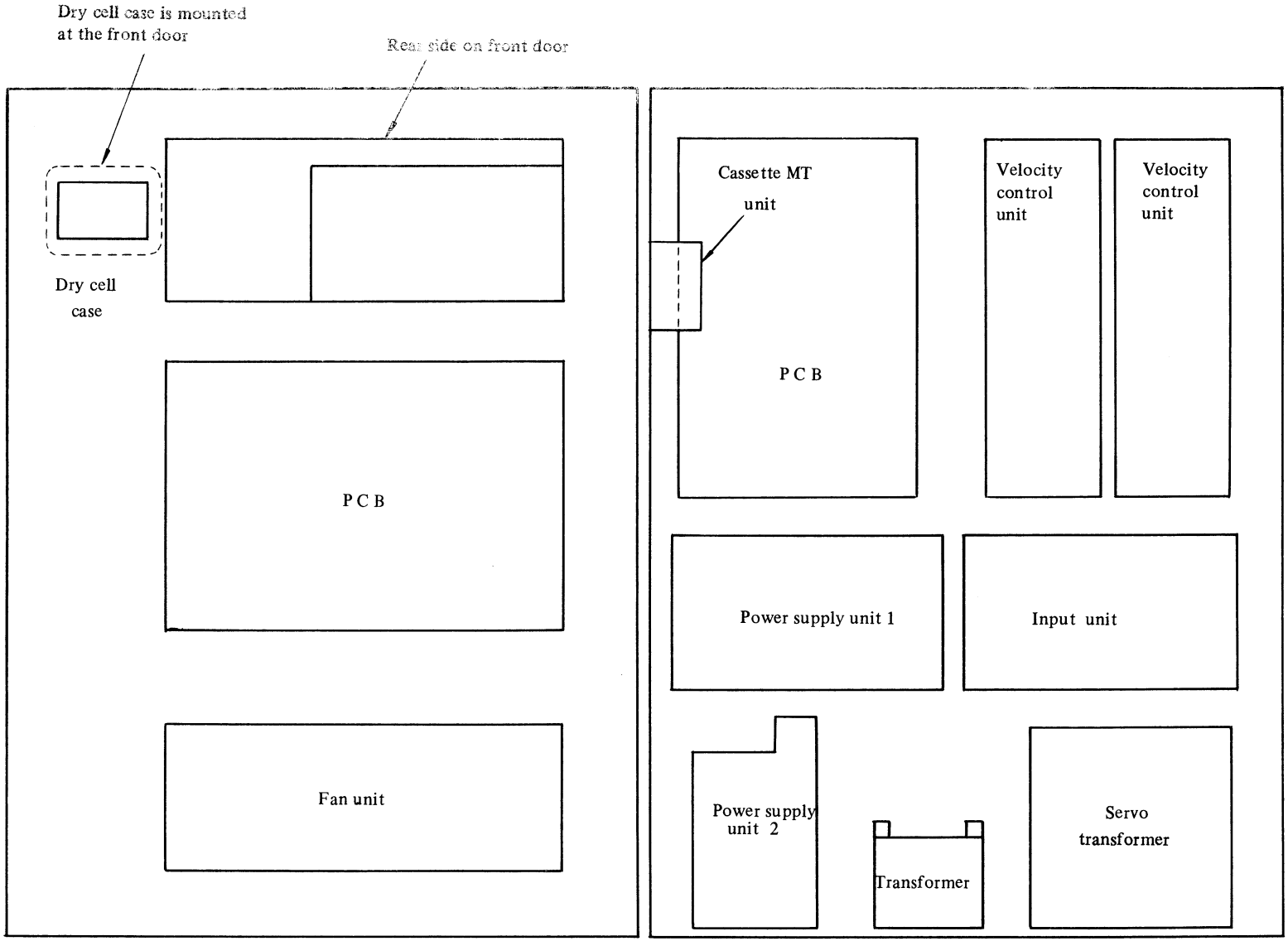
1. MOUNTING LAYOUT OF UNITS IN MAIN BODY CABINET

Fig. 1.1 to 1.3 indicate the mounting layout of units in the main body cabient of NC.



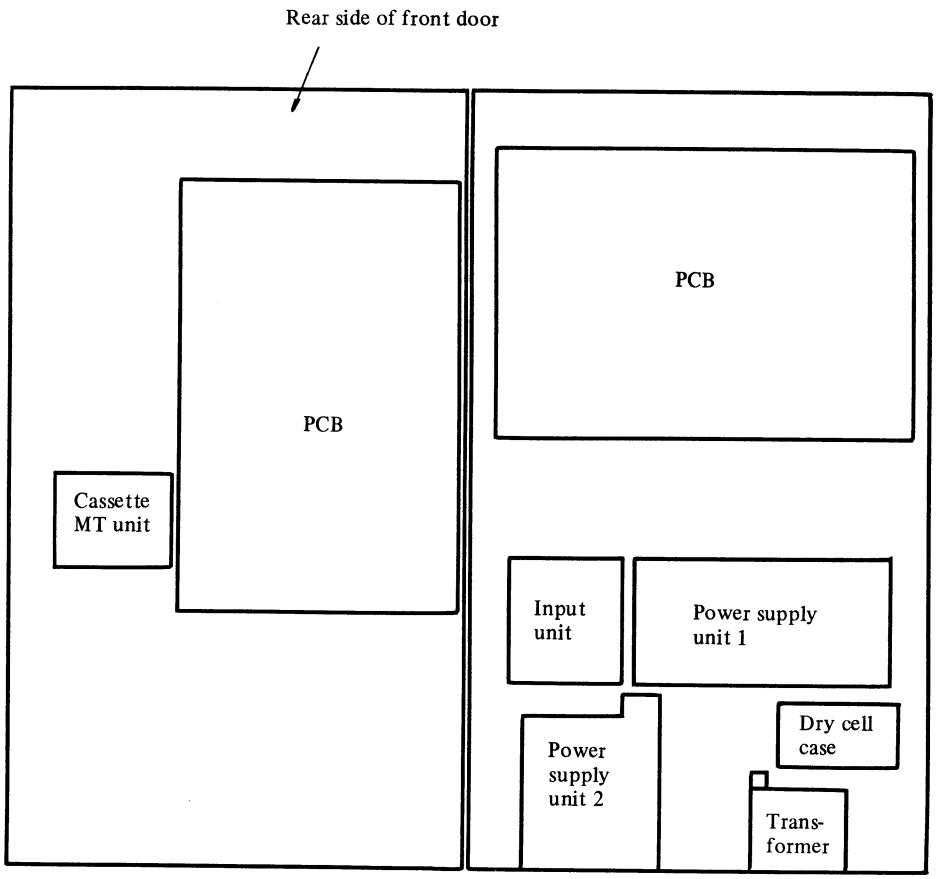
Front view with the front door of cabinet open

Fig. 1.1 Built-in type 1 cabinet



Front view with the front door of cabinet open.

Fig. 1.2 Built-in type 2 cabinet



Front view with the front door of cabinet open

Fig. 1.3 Unbundled type cabinet

2. PERIODIC MAINTENANCE

2.1 Dry Cell Exchange

Replace dry cells once every year so as to hold the contents of data memory in NC after power supply has been turned off. For the mounting position of the dry cell case, refer to Figs. 1.1 to 1.3.

(1) Replace dry cells when the NC power supply remains turned on.
(2) Be careful with polarity of dry cells so as not to insert them reversely.
(3) Use alkali manganese dry cells 1 X 3 pcs. being available in the market as exchange dry cells.

2.2 Cleaning of Cassette MT Unit

This NC loads Symbolic FAPT system programs in cassette magnetics tape. When the LOAD key of the MDI & CRT unit is depressed, data of the cassette MT are loaded to the internal memory of NC, and the Symbolic FAPT processing is done according to the NC internal memory data.

The cassette MT is mounted inside the NC cabinet, and it does not require any operation usually. However, if the surface of the cassette MT head is contaminated with dust or the like, the reliability of loaded data extremely deteriorates.

Accordingly, clean the cassette MT unit periodically according to the following procedure.

For the mounting position of the cassette MT inside NC cabinet, refer to Figs. 1.1 to 1.3.

2.2.1 Names of component parts of cassette MT

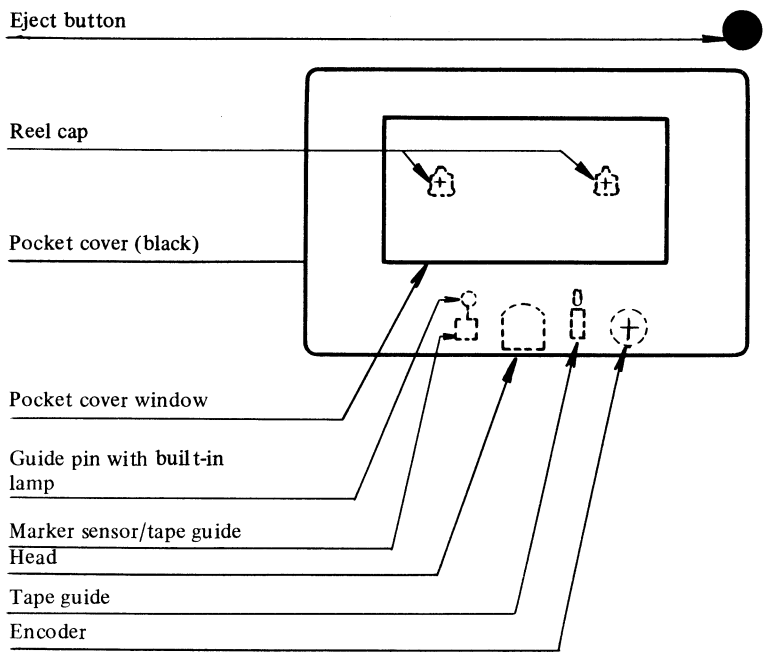


Fig. 2.1 Front view

Fig. 2.2 indicates that the cassette has been loaded after opening the pocket cover by depressing the eject button.

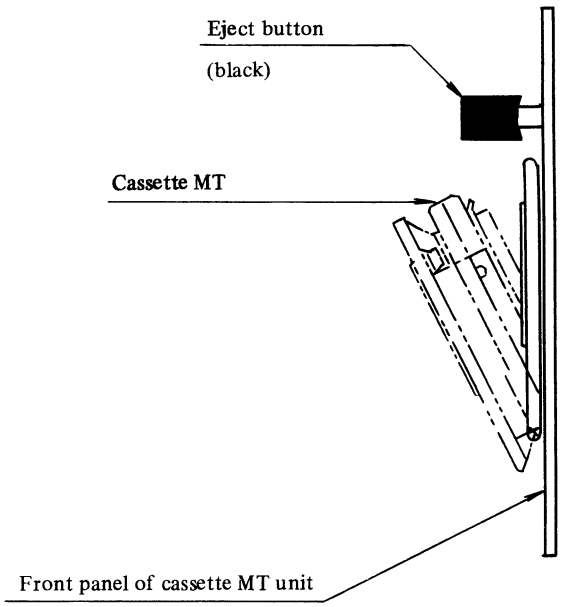


Fig. 2.2 Right side view

2.2.2 Cleaning

(1) Cleaning interval

Clean the cassette MT unit once every 400 times of Symbolic FAPT system program loading or once every 3 months.

(2) Cleaning tools

(a) Wool cloth free of waste pieces of thread, like gauze or cotton applicator.

(b) Trichloro-ethane cleaner solvent

The following devices are attached as exclusive cleaning devices

Cleaner kit: A87L-0001-0040 #TZ-350 (TEAC's TZ-350 cleaner kit)
consisting of cotton applicators, gauze, and cleaner solvent

(3) Cleaning procedure

(a) Remove the pocket cover. Unload the tape, in advance. Detach the pocket cover by pulling its lower latch part in the left direction of the arrow in Fig. 2.3 Ⓑ , which depressing the entire pocket cover toward the arrow direction (head direction) in Fig. 2.3 Ⓐ .

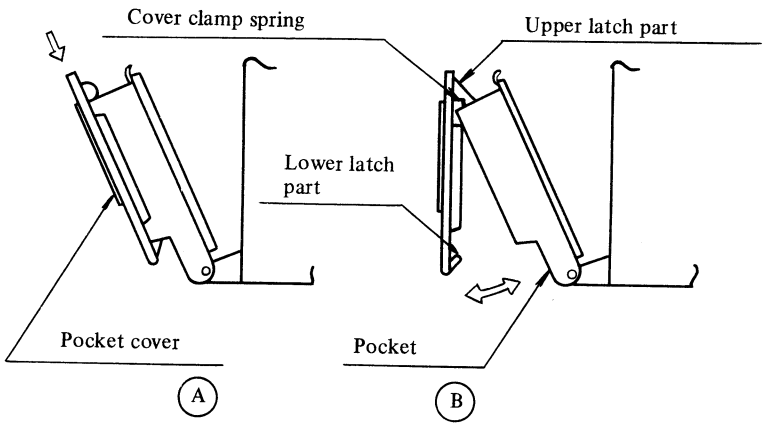


Fig. 2.3 Detaching method of pocket cover

(b) Clean the head surface carefully by using the cotton applicator or cloth wetted with cleaner solvent.
Clean two tape guides and tape contact part of the encoder with due care.
Since these tape guides and tape contact part of encoder are not easily contaminated with dust as compared with the head, clean them once every 2 to 3 cleaning times of the head.
Be careful not to wet the cotton applicator or cloth with cleaner solvent excessively.
Carefully clean the encoder and left tape guide/marker sensor slowly and carefully without applying a force.
Clean the ecoder, while turning it by hand.

(c) Wipe two guide pins and other mechanical part surfaces with a dry cloth. Don't use cleaner solvent.
Be careful not to leave the waste pieces of thread or the like in the lamp window of the left guide pin (with built-in lamp).

(d) Mount the pocket cover.
Hook the upper latch of the cover to the cover clamp spings mounted on both sides of the tape loading port of the pocket. (See Fig. 2.3 Ⓑ)

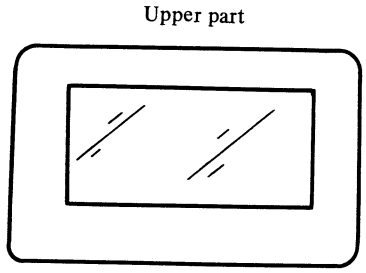


Fig. 2.4 Pocket cover

Fit the pocket cover to the pocket by depressing it in the arrow direction (pocket direction) in Fig. 2.3 Ⓑ , while depressing the entire cover in the head direction.

3. MELTING AND EXCHANGE FUSES

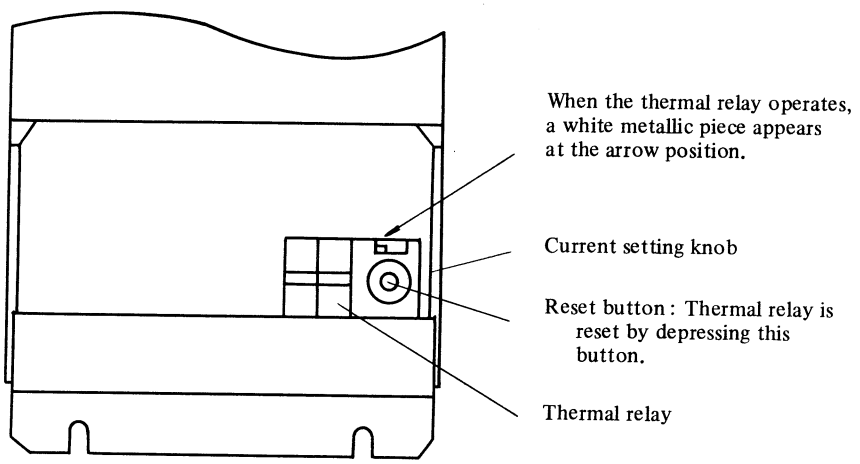
If a fuse of the NC was blown out, locate and eliminate its cause, and then, replace the blown out fuse.

3.1 Fuses for Control Unit

Mounting place	Name of fuse	Capacity	Type	Uses
Input unit	F1, F2	10A	Cellolite fuse	Control unit input power supply
	F3	0.32A	Alarm fuse (small size)	Power ON/OFF control circuit
Power supply unit 1	AF11, AF12	5A	Alarm fuse	Power supply unit AC input
Power supply unit 2	F1	1A	〃	〃
	F2	3.2A	〃	
MDI & CRT unit		2.5A		CRT unit power supply

3.2 Thermal Relay of Velocity Control Unit

A thermal relay is mounted in the velocity control unit to operate if the velocity control unit is overloaded.



4. CHECK AND CLEANING OF MOTOR BRUSH

Check and clean the motor brush in the way explained in the following. If the motor brush is abnormally work because of forgetting the check, the motor can be damaged as the result, therefore, be sure to check the motor brush.

(1) Periodic check should be made at the intervals listed in the following as the standard.

- In the case of a general machine tool (lathe, milling machine, machining center, or such): Every one year
- In the case of a machine tool with a high frequency of acceleration/deceleration (turret punch press or such): Every two months

  However, it is recommended that the check interval should be determined judging the actual wear situation of the motor brush.

(2) Confirm that the power supply to the DC servo motor (machine) is OFF. Immediately after the DC servo motor has been operated, the brush may be hot. In such a case, make the check after the brush is completely cooled.

(3) Remove the brush cap, as shown in Fig. A, using a screwdriver which fits to the slot.

(4) After taking out the brush completely, measure (visually) the length of the brush (see Fig. B). If the length of the remaining brush is shorter than 10 mm (in the case of motor model 00M, 5 mm), the brush cannot be used any more.

Taking this fact into consideration, make a judgement as to whether the brush can be used until the next check time, and if necessary, replace the brush with a new one.

(5) Check the brush very carefully. If any deep groove or scar is found on the contact surface of the brush or if any mark of arcing is perceived on the brush spring, replace the brush with a new one. In this case, check the brush occasionally for about a month after the replacement, and if the same situation happens during this period, contact our nearest service station.

(6) Blow off the brush dust in every brush holder with compressed air (factory air), and the brush dust will come out through another brush holder. Before using the compressed air, confirm that the air does not contain iron dust or a large amount of moisture.

(7) After the check, put back the brush and tighten the brush cap fully. In this case, be careful that sometimes the brush spring is caught in between the conducting metal and brush holder and the brush cap cannot go as far as the depth. Confirm that all the brush caps are tighten into the respective brush holders to almost the same level. When putting the brush into the brush holder, sometimes the brush cannot smoothly slide due to the brush dust which adhered to the inside surface of the brush holder. In such a case, clean the inside surface of the brush holder with the tip of a screwdriver. (Take care not scratch the commutator surface.)

(8) When replacing the brush, use just the same brush (in the quality, shape, etc.) as the existing one. After replacement of the brush, run the DC servo motor without load for a while to fit the brush surface to the commutator surface.

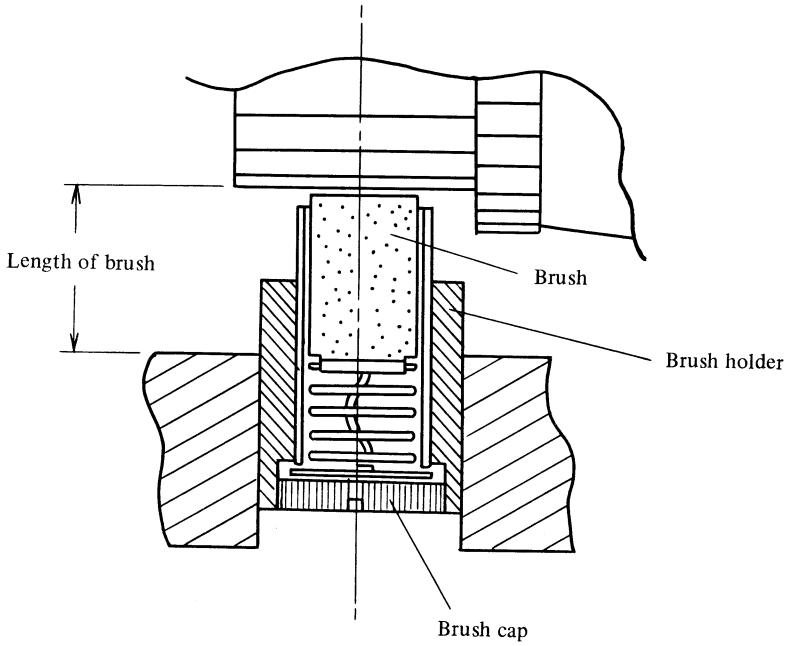
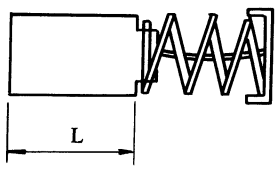


Fig. A Structure of Brush Holder



Motor model	Length of new brush	Usable length
Model 00M	10 mm	5 mm
Model 0M, 5M	19 mm	10 mm
Model 10M, 20M, 30M	19 mm	10 mm

Fig. B Brush Length

5. DISPLAY OF NC STATES

When the NC do not operate without alarm state, the states of the NC can be inspected by lighting DGNOS LED and setting 700 or 701 on the number indicator.

	7	0	0

	CSCT	CITL	COVZ	CINP	CDWL	CMTN	CFIN
OH	MEMORY	OT	SV		P/S		
7	6	5	4	3	2	1	0

Meaning of each digit where '1' is displayed are as follows;

CFIN: M, S or T function has been executed.

CMTN: Move command in automatic operation has been executed.

CDWL: Dwell command has been executed.

CINP: Inposition check has been performed.

COVZ: Override dial has been set to zero position.

CITL: Inter lock signal (STLK) has been turned on.

CSCT: The NC waits for turning on the spindle speed arrival signal.

	7	0	1

		CRST				CTRD	CTPU
OH	MEMORY	OT	SV			P/S	
7	6	5	4	3	2	1	0

CTPU: memory has output via the RS 232C interface.

CTRD: The program tape has been read via RS232C interface.

CRST: Emergency stop button, remote reset button or reset button on MDI panel is turned on.

	7	1	2

STP	REST	EMS		RSTB			CSU
OH	MEMORY	OT	SV			P/S	

The state during cycle operation stop or cycle operation pause is confirmed. This is used for troubleshooting.

STP: The flag which stops the pulse distribution.
This is set at the following condition.
(a) External reset has been turned on.
(b) Emergency stop has been turned on.
(c) Feed hold has been turned on.
(d) Reset button on the MDI & DPL panel is on.
(e) The mode has been changed to the manual mode. (JOG, HANDLE/STEP)
(f) Alarm has been generated. (some alarms may not set the flag.)

REST: This is set when one of the external reset, emergency stop, or reset button has been turned on.

EMS: This is set when the emergency stop has been set on.

RSTB: This is set when the reset button is on.

CSU: This is set when the emergency stop has been turned on or when the servo alarm has been generated.

	8	0	0	SVERRX (X axis)
	8	0	1	SVERRZ (Z axis)

Position deviation value of X and Z axes in order.

	8	2	0	ABSMTX (X axis)
	8	2	1	ABSMTZ (Z axis)

Machine position from the reference point of X and Z axes in order.

APPENDIX

APPENDIX 1 G FUNCTION TABLE

The following G codes are available. The optional special G code can also be used.

Standard G code	Group	Function	Classification
◤G00	01	Positioning	B
G01		Linear interpolation	B
G02		Circular interpolation (CW)	B
G03		Circular interpolation (CCW)	B
G04	00	Dwell	B
G10		Offset value setting	O
G20	06	Inch data input	O
G21		Metric data input	O
G27	00	Reference point return check	B
G28		Return to reference point	B
G32	01	Thread cutting	B
G50	00	Programming of absolute zero point, setting of max. spindle speed	B, O
G68	04	X axis mirror image ON	O
◤G69		Mirror image cancel	B
G70	00	Finishing cycle	B
G71		Stock removal in turning	B
G72		Stock removal in facing	B
G73		Pattern repeating	B
G74		Peck drilling in Z-axis	B
G75		Grooving in X-axis	B
G76		Thread cutting cycle	B
G90	01	Cutting cycle A	B
G92		Thread cutting cycle	B
G94		Cutting cycle B	B
G96	02	Constant surface speed control	O
◤G97		Constant surface speed control cancel	B
G98	05	Feed per minute	B
◤G99		Feed per revolution	B

(Note 1) The G codes marked with ◤ are initial G codes in each group. That is, when turning power on, this G code is set.

(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. (Alarm number 010)*

(Note 4) *A number of G codes can be commanded in a block 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) *A G code from each group is displayed.*

APPENDIX 2 TABLE OF COMMAND VALUE

		Input mm Output mm	Input inch Output mm	Input mm Output inch	Input inch Output inch
Least input increment		0.001 mm	0.0001 inch	0.001 mm	0.0001 inch
Max. stroke (distance from the reference point)		±9999.999 mm	±9999.999 mm	±999.9999 inch	±999.9999 inch
Max. programmable dimension		±9999.999 mm	±999.9999 inch	±9999.999 mm	±999.9999 inch
Cutting feed rate (at override 100%)	Feed per minute	1 ~ 15000 mm/min	0.01 ~ 600.00 inch/min	1 ~ 15000 mm/min	0.01 ~ 600.00 inch/min
	Feed per revolution	0.0001 ~ 500.0000mm/rev	0.000001 ~ 9.999999 inch/rev	0.0001 ~ 500.0000mm/rev	0.000001 ~ 9.999999 inch/rev
Rapid traverse feed rate (separate for each axis)		30 ~ 15000 mm/min	30 ~ 15000 mm/min	3.0 ~ 600.0 inch/min	3.0 ~ 600.0 inch/min
Upper limit of cutting feed rate		6 ~ 15000 mm/min	6 ~ 15000 mm/min	0.6 ~ 600.0 inch/min	0.6 ~ 600.0 inch/min
Fo speed					
Manual jog feed rate		0 ~ 1260 mm/min	0 ~ 49.6 inch/min	0 ~ 1270 mm/min	0 ~ 50.0 inch/min
Thread lead		0.0001 ~ 500.0000 mm	0.000001 ~ 9.999999 inch	0.0001 ~ 500.0000 mm	0.000001 ~ 9.999999 inch
Max. spindle speed		9999 rpm	9999 rpm	9999 rpm	9999 rpm
Tool offset amount		0 ~ ±999.999 mm	0 ~ ±99.9999 inch	0 ~ ±999.999 mm	0 ~ ±99.9999 inch
Step amount in incremental feed		0.001 mm	0.0001 inch	0.001 mm	0.0001 inch
Backlash compensation amount		0 ~ 0.255 mm	0 ~ 0.255 mm	0 ~ 0.0255 inch	0 ~ 0.0255 inch
Area of stored stroke limit (value from the reference point)		±9999.999 mm	±9999.999 mm	±999.9999 inch	±999.9999 inch
Dwell		0 ~ 9999.999 sec	0 ~ 9999.999 sec	0 ~ 9999.999 sec	0 ~ 9999.999 sec.

APPENDIX 3 PARAMETER LIST

Parameters not mentioned here should be set to zero.

		0	4

ADFT	RDRN	DECI	ORC	TOC	DCS	PROD	SW
OH	MEMORY	OT	SV		P/S		
7	6	5	4	3	2	1	0

ADFT
1: Automatic drift compensation is performed.
0: Automatic drift compensation is not performed.

RDRN
1: Dry run is effective for the rapid traverse command
0: Dry run is ineffective for the rapid traverse command.

DECI
1: Deceleration signal '1' in reference point return shows deceleration.
0: Deceleration signal '0' in reference point return shows decelration.

ORC
1: Offset value becomes a radius designation.
0: Offset value becomes a diameter designation.

TOC
1: Offset is canceled in reset state.
0: Offset is not canceled in reset state.

DCS
1: Pushing the START button on the MDI panel directly actuates the NC start without going through the machine side. (MDI mode only).
0: Pushing the START button on the MDI panel issues the signal to the machine side. The NC start is actuated when the NC receives the start signal from machine side.

PROD
1: The programmed position is displayed on the position display of address U or W.
0: The position in which the tool offset is considered is displayed on the position display.

SCW
1: Least command increment is that in inch system (machine tool: inch system).
0: Least command increment is that in metric system (machine tool: metric system).

		0	5

NFED					ASR33	PPD	STP2
OH	MEMORY	OT	SV		P/S		
7	6	5	4	3	2	1	0

NFED
1: Feed is not output at program output by I/O interface.
0: Feed is output at program output by I/O interface.
(This parameter is effective, when the setting parameter "I/O" is zero.)

ASR33
1: 20 mA current interface is used as I/O interface.
0: RS232C is used as I/O interface.
(This parameter is effective, when the I/O is zero.)

PPD
1: Relative coordinate values are preset by programming of absolute zero point (G50 or G92).
0: Relative coordinate values are not preset by programming of absolute zero point (G50 or G92).

STP2
1: 2 bits are taken as the stop bit, when using the RS232C interface.
0: 1 bit is taken as the stop bit.

(This parameter is effective, when the I/O is zero)

		0	6

PSG2	PSG1		OVR1			ZMZ	ZMX
OH	MEMORY	OT	SV		P/S		
7	6	5	4	3	2	1	0

PSG2, PSG1 Gear ratio of spindle and position coder

Gear ratio of spindle and position coder

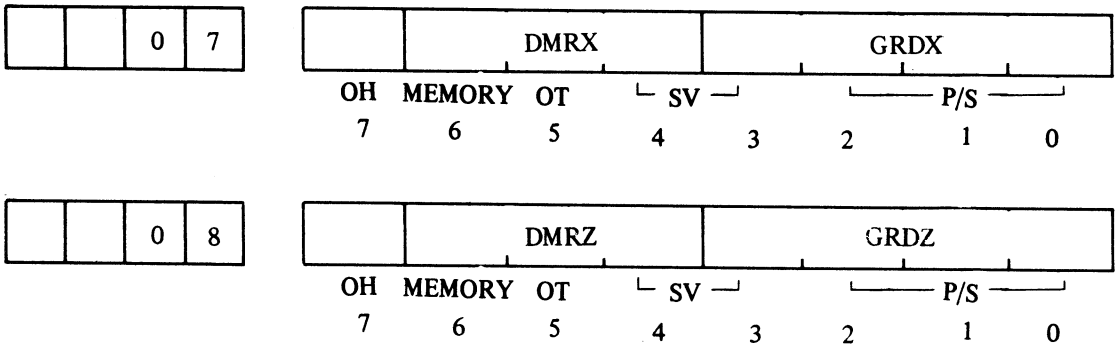
Magnification	PSG2	PSG1
x 1	0	0
x 2	0	1
x 4	1	0
x 8	1	1

$$\text{Magnification} = \frac{\text{Number of spindle rotation}}{\text{Number of position coder rotaiton}}$$

OVR1
1: The polarity of override signal (*OV1 to *OV8, RV1, and RV2) Feedrate becomes fast when override signal is "1".
0: The polarity of override signal (*OV1 to * OV8, RV1, and RV2) Feedrate becomes fast when override signal is "0".

ZMX, ZMZ X axis or Z axis reference point return direction in order, and the initial backlash direction when turning on the power.
1: Reference point return direction and the initial backlash direction are minus.
0: Reference point return direction and the initial backlash direction are plus.

(Note) *The backlash compensation is initially performed when the axis moves in the positive direction against the direction which is set by this parameter after the power is turned on.*



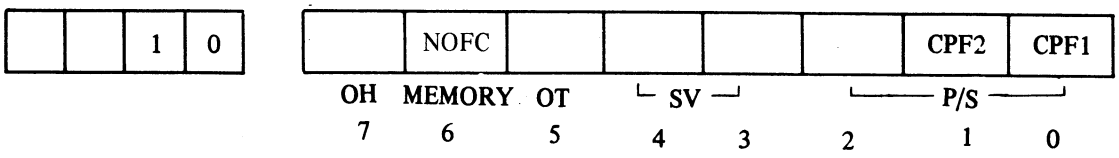
DMRX, DMRZ Command multipling ratio for X and Z axis in order.

Setting code			Multipling ratio
6	5	4	
0	0	0	1/2
0	0	1	1
0	1	0	1
0	1	1	2
1	0	0	3/2
1	0	1	3
1	1	0	2
1	1	1	4

GRDX, GRDZ Capacity of reference counter for X and Z axis in order.

Setting code				One cycle capacity
3	2	1	0	
0	0	0	1	2000
0	0	1	0	3000
0	0	1	1	4000
0	1	0	0	5000
0	1	0	1	6000
0	1	1	1	8000
1	0	0	1	10000

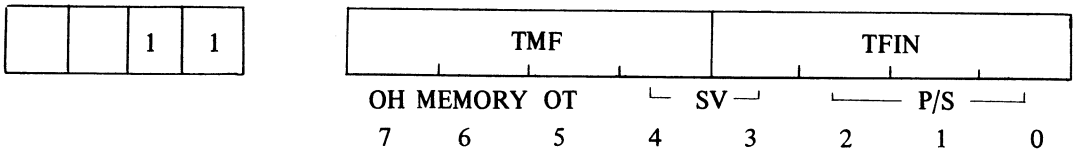
(Note) *If the code other than codes in the above table is set, capacity is set 8000.*



CPF2, CPF1 Backlash compensation pulse frequency (common to all axes.)

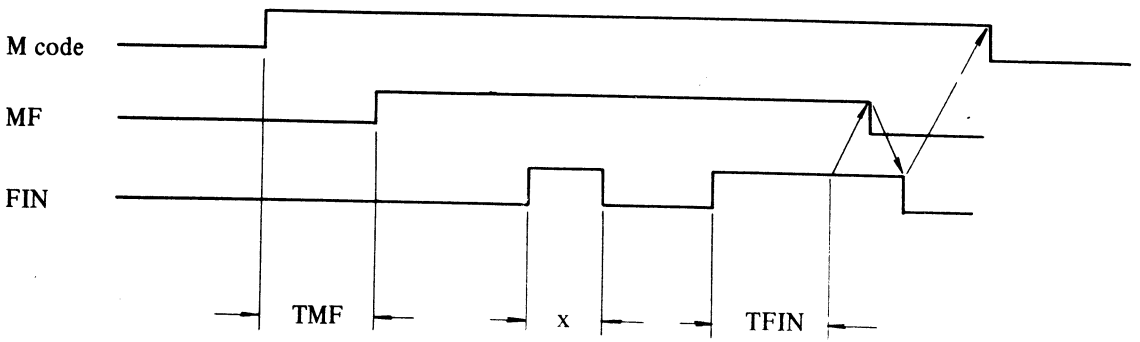
Frequency KHz	CPF2	CPF1
32	0	0
64	0	1
128	1	0
256	1	1

NOFC
1: Counter input of offset amount is ineffective.
0: Counter input of offset amount is effective.



TMF Time from M, S, T code issue to MF, SF, TF issue
16 ~ 256 m sec. (16 m sec. increment).

TFIN Time of reception width of FIN
16 ~ 256 m sec (16 m sec. increment).



Because of x < TFIN, it is neglected.

TMF	TFIN	Parameter setting			
16 (msec)	More than 16 (msec)	0	0	0	0
32	" 32	0	0	0	1
48	" 48	0	0	1	0
64	" 64	0	0	1	1
80	" 80	0	1	0	0
96	" 96	0	1	0	1
112	" 112	0	1	1	0
128	" 128	0	1	1	1
144	" 144	1	0	0	0
160	" 160	1	0	0	1
176	" 176	1	0	1	0
192	" 192	1	0	1	1
208	" 208	1	1	0	0
224	" 224	1	1	0	1
240	" 240	1	1	1	0
256	" 256	1	1	1	1

		1	2

APRS	WSFT	DOFSI			OFFVY		ISOT
OH	MEMORY	OT	SV		P/S		
7	6	5	4	3	2	1	0

APRS
1: Automatic coordinate system setting is performed at manual reference point return.
0: Automatic coordinate system setting is not performed.

WSFT
1: The work coordinate system is shifted, when the setting amount on this function is set to the offset number 0 or 100.
0: The work coordinate system shift is not performed.

DOFSI
1: Direct measured value input for tool offset is performed by setting to the offset number 101 to 116.
0: Direct measured value input for tool offset is not performed.

OFFVY
1: The servo alarm is not generated, even if VRDY signal becomes ON before PRDY signal is output.
0: The servo alarm is generated, when VRDY signal becomes ON before PRDY signal is output.

ISOT
1: The rapid traverse is effective without reference point return after turning power on or emergency stop state.
0: The rapid traverse does not become effective until reference point return is performed after turning power on or emergency stop state.

		1	3

MCINP							
OH	MEMORY	OT	SV		P/S		
7	6	5	4	3	2	1	0

MCINP
1: Program can be started by external signal (MINP).
0: Program cannot be started by external signal (MINP).

		1	4

NFED					ASR33		STP2
OH	MEMORY	OT	SV		P/S		
7	6	5	4	3		1	0

NFED
1: Feed is not output at program output by I/O interface.
0: Feed is output at program output by I/O interface.
(This parameter is effective, when the setting parameter "I/O" is 1.)

ASR33
1: 20 mA current interface is used as I/O interface.
0: RS232C is used as I/O interface.
(This parameter is effective, when "I/O" is 1.)

STP2
1: 2 bits are taken as the stop bit, when using I/O interface.
0: 1 bit is taken as the stop bit, when using I/O interface.
(This parameter is effective, when "I/O" is 1.)

The following parameters are set by decimal number.

		1	5	CMRX

		1	6	CMRZ

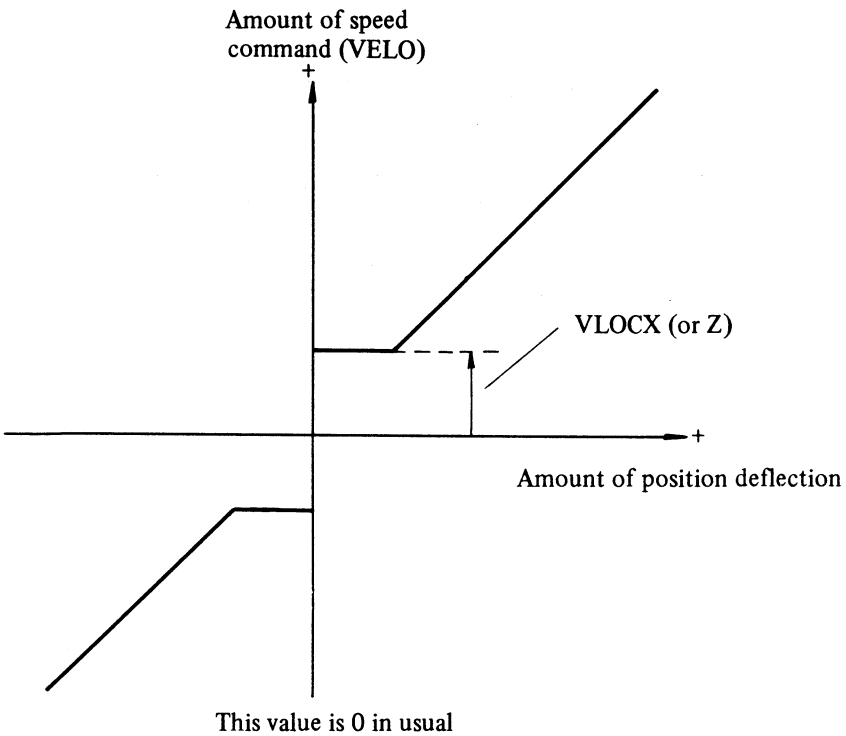
CMRX, CMRZ Command multiply for each axis

Setting code	Multiply
1	0.5
2	1
4	2
10	5
20	10

		1	8	VLOCX

		1	9	VLOCZ

VLOCX, VLOCZ Clamp of feed command value of each axis.
Setting amount 0 ~ 7 VELO.



		2	1	SPLOW

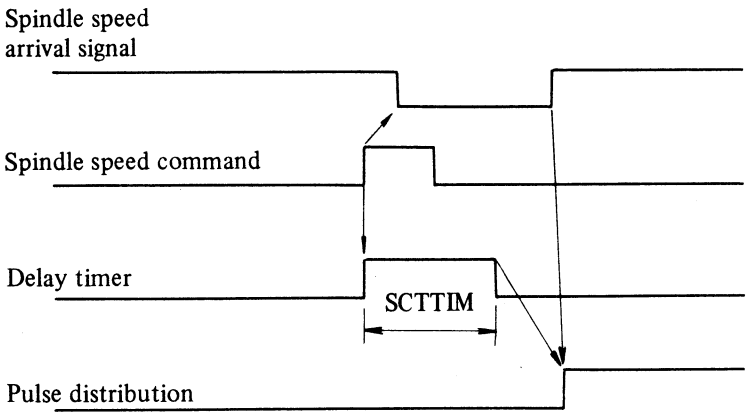
SPLOW Spindle speed during spindle low rotation
setting amount 0 ~ 255 Unit: rpm

		2	2	THDCH

THDCH Width of chamfering for thread cutting cycle (G92, G76)
Setting amount 0 ~ 127 Unit: 0.1 lead

		2	3	SCTTIM

SCTTIM — Delay time interval in spindle speed arrival signal check. Set the time interval from spindle function execution to check of spindle speed arrival signal.
Setting amount 0 ~ 255 Unit: msec.



		2	5	INPX
		2	6	INPZ

INPX, INPZ — Width of X or Z axis in-position in order.
Setting amount 0 ~ 32767 detection unit

		2	8	SERRX
		2	9	SERRZ

SERRX, SERRZ — The position loop error limit for X or Z axis in order.
Setting amount 0 ~ 32767 detection unit.

		3	1	GRDSX
		3	2	GRDSZ

GRDSX, GRDSZ — Setting of grid shift amount of X or Z axis in order.
Setting amount 0 ~ ±32767 detection unit.
The sign of parameter means the shifting direction of reference point.

		3	4	LPGMX

		3	5	LPGMZ

LPGMX, LPGMZ

Setting of servo loop gain multiple of X or Z axis in order.

$$\text{Setting amount} = 2048 \times \frac{E}{L} \times \alpha \times 1000$$

$$E = \begin{cases} 7 \text{ [V] (For motor with 7V at 1000 rpm)} \\ 3.5 \text{ [V] (For motor with 7V at 2000 rpm)} \end{cases}$$

L: Machine movement amount per motor revolution (mm or inch)

α: Detect unit (mm or inch)

(Example) 2 mm per motor revolution at 1000 rpm/7V.

$$\text{Setting value: } 2048 \times \frac{7}{2} \times 0.0005 \times 1000 = 3584$$

(at detect unit 0.0005 mm)

Setting value should be rounded at the decimal point.

		3	6	PSANGN

PSANGN

Data of adjustment for gain in constant surface speed control (S analog output)
Set the data of adjustment for gain in spindle analog output data.
Setting amount: 700 ~ 1250
Standard amount: 1000

[Adjustment method]

(1) Set standard value "1000".
(2) Command maximum value in S analog code. (10V)
(3) Measure output voltage.
(4) Reset PSANGN according to below formula.

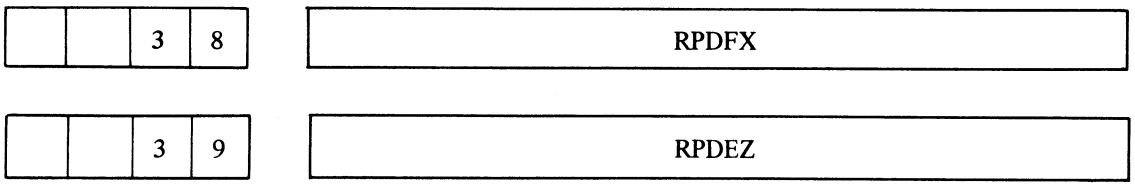
$$\frac{10.00}{\text{Measuring Voltage}} \times 1000 = \text{Setting Value}$$

(5) After setting new data, check whether output voltage is maximum voltage (10V).

		3	7	LPGIN

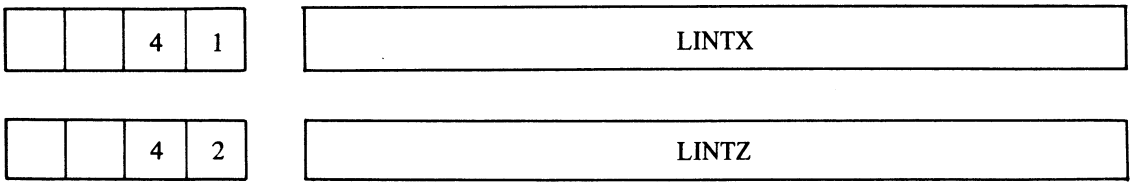
LPGIN

Setting of position control loop gain.
Setting amount 1 ~ 9999 Unit: 0.01 sec^{-1}



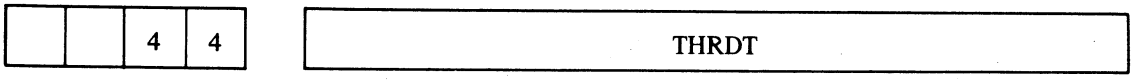
RPDFX, RPDFZ

Rapid traverse rate of X or Z axis in order.
Setting amount 30 ~ 15000 Unit: mm/min (mm output)
30 ~ 6000 Unit: 0.1 inch/min (inch output)



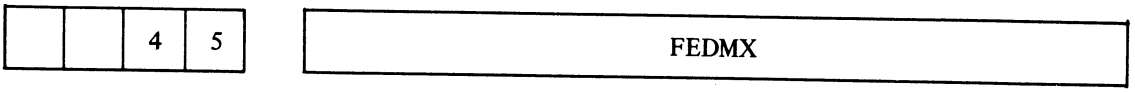
LINTX, LINTZ

The time constant value in linear acceleration/deceleration of X or Z axis in order (for rapid traverse).
Setting amount 8 ~ 4000 (unit: msec.)



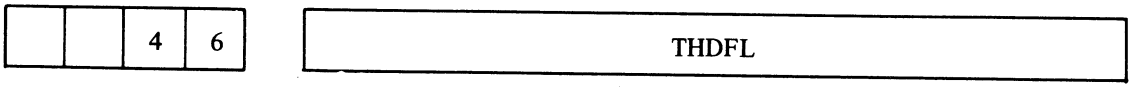
THRDT

The time constant value of X axis in thread cutting cycle G92 (G78)
Setting amount 1 ~ 4000 (unit: msec.).
The most suitable value should be set with this parameter and THDFL (parameter number 46).



FEDMX

Upper speed of cutting feed. (Available for all axis)
Setting amount 6 ~ 15000 Unit: mm/min (mm output)
6 ~ 6000 Unit: 0.1 inch/min (inch output)



THDFL

The lower limit value of X axis acceleration/deceleration in thread cutting cycle G92 (G78). That is FL speed, and common to all axes.
Setting amount: 6 ~ 15000 Unit: mm/min (mm output)
6 ~ 6000 Unit: 0.1 inch/min (inch output)
The most suitable value should be set with this parameter and THRDT (parameter number 44).

		4	7	FEEDT

FEEDT

Time constant of exponential acceleration/deceleration of feed and jog feed.
Setting amount 0 ~ 4000 Unit: msec.
When the exponential acceleration/deceleration is not effective, set '0' to this parameter.

		4	8	FEDFL

FEDFL

Low speed of exponential acceleration/deceleration of feed (FL speed)
Setting amount 6 ~ 15000 Unit: mm/min (mm output)
6 ~ 6000 Unit: 0.1 inch/min (inch output)

		5	1	RPDFL

RPDFL

Least speed of rapid traverse override (Fo) common to all axis.
Setting amount 6 ~ 15000 Unit: mm/min (mm output)
6 ~ 6000 Unit: 0.1 inch/min (inch output).

		5	2	ZRNFL

ZRNFL

LOw feed speed at reference point return, FL speed. (Common to all axes).
Setting amount 6 ~ 15000 Unit: mm/min (mm output).
6 ~ 6000 Unit: 0.1 inch/min (inch output).

		5	3	BKLX
		5	4	BKLZ

BKLX, BKLZ

Backlash amount of X or Z axis in order.
Setting amount 0 ~ 255 Unit: 0.001 mm (mm output).
0 ~ 255 Unit: 0.0001 inch (inch output)

(Setting amount on X axis should be diameter value, in diameter programming)

		5	6	SPDLC

SPDLC

Set spindle speed offset compensation value, that is, compensation value of zero offset of spindle speed command voltage.
Setting amount 0 ~ ±8191 (Unit: VELO)

(It is used for optional function, constant surface speed control.)

		5	7	GRMX 1
		5	8	GRMX 2
		5	9	GRMX 3
		6	0	GRMX 4

GRMX1 ~ GRMX4 Spindle speed corresponding with gear 1 ~ 4, when spindle speed command is 10V.
Setting amount 1 ~ 9999 (Unit: rpm)

(It is used for optional function, constant surface speed control.)

		6	1	DRFTX
		6	2	DRFTZ

DRFTX, DRFTZ Compensation amount of drift generated in servo loop of X or Z axis in order.
Setting amount 0 ~ ±8191 (Unit: VELO)

		6	4	JOGFL

JOGFL Low speed of exponential acceleration/deceleration of jog feed (FL speed)
Setting amount 6 ~ 15000 Unit: mm/min (mm output)
6 ~ 6000 Unit: 0.1 inch/min (inch output)

		6	7	LOWSP

LOWSP Least spindle speed in constant surface speed control mode (G96).
Setting amount 0 ~ 9999 (Unit: rpm).

		6	8	BRATE 0

BRATE 0 Baud rate setting in using I/O interface
Setting amount 50 ~ 4800 (Unit: BPS)
(This parameter is effective, when "I/O" is 0.)
(50, 100, 110, 150, 200, 300, 600, 1200, 2400, 4800)

		6	9	BRATE 1

BRATE 1

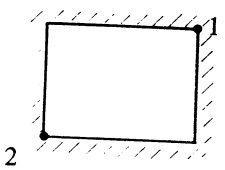
Baud rate setting in using I/O interface
Setting amount 50 ~ 4800 (Unit: BPS)
(This parameter is effective, when "I/O" is 1.)
(50, 100, 110, 150, 200, 300, 600, 1200, 2400, 4800)

		7	0	LT1X1
		7	1	LT1Z1
		7	3	LT1X2
		7	4	LT1Z2

LT1 □ □

→ nth top in square zone (See figure)

→ axis



Set stroke limit mentioned above.
Setting amount 0 ~ ±9999999 (Unit: 0.001 mm or 0.0001 inch)
Set the distance from the reference point to the limit.
In case of diameter command, X axis is set by diameter amount.
Outside the area specified by parameters is forbidden area.
Generally, stored stroke limit should be set at the maximum stroke. If the machine movable part enters into the forbidden area, the overtravel alarm is generated.

		7	6	PRSX
		7	7	PRSZ

PRSX, PRSZ

The coordinate values of the reference point return in X and Z axis in automatic coordinate system setting.

Setting amount 0 ~ ± 9999999
Unit 0.001 mm (metric input) or 0.0001 inch. (inch input)

Set the PRSX in diameter value in diameter designation.

		8	2	MRCCD

MRCCD

Cutting depth in multiple repetitive cycle (G71, G72)
Setting amount 1 ~ 9999999 Unit: 0.001 mm (mm input)
1 ~ 9999999 Unit: 0.0001 inch (inch input)

		8	3	MRCDT

MRCDT

Relief amount in multiple repetitive cycle (G71, G72)
Setting amount 1 ~ 9999999 Unit: 0.001 mm (mm input)
1 ~ 9999999 Unit: 0.0001 inch (inch input)

		8	4	PESCX
		8	5	PESCZ

PESCX, PESCZ

Relief amount in multiple repetitive cycle (G73) in X and Z axis direction.
Setting amount 0 ~ ±9999999 Unit: 0.001 mm (mm input)
0 ~ ±9999999 Unit: 0.0001 inch (inch input)

		8	6	PATIM

PATIM

Number of division in multiple repetitive cycle (G73)
Setting amount 1 ~ 9999999

		8	7	GROVE

GROVE

Return amount in multiple repetitive cycle (G74, G75)
Setting amount 0 ~ 9999999 Unit: 0.001 mm (mm input)
0 ~ 9999999 Unit: 0.0001 inch (inch input)

		8	8	THRPT

THRPT

Repetitive count of finishing in multiple repetitive cycle (G76)
Setting amount 1 ~ 9999999

		8	9	THANG

THANG

Angle of tool tip in multiple repetitive cycle (G76)
Setting amount 0, 29, 30, 55, 60, 80

		9	0	THCLM

THCLM

Minimum cutting depth in multiple repetitive cycle (G76)
Setting amount 0 ~ 9999999 Unit: 0.001 mm (mm input)
0 ~ 9999999 Unit: 0.0001 inch (inch input)

		9	1	THDFN

THDFN

Finishing allowance in multiple repetitive cycle (G76)
Setting amount 0 ~ 9999999 Unit: 0.001 mm (mm input)
0 ~ 9999999 Unit: 0.0001 inch (inch input)

APPENDIX 4 ALARM LIST

(1) Overheat Alarm (OH)

No.	Content	Remarks
1	Printed Circuit Board is overheated.	
2	DC motor is overheated.	
3	Both P.C.B. and DC motor are overheated.	

(2) Memory Alarm (MEMORY)

No.	Content	Remarks
1	RAM Parity Error (low bite)	
2	RAM Parity Error (high bite)	
3	ROM Parity Error.	
4	ROM Pair Error (No correspondence between high and low)	
5	C-MOS Parity Error	
6	System Error (Watch dog timer alarm)	
7	CPU Error (0, 3, 4 type interrupt generation)	

(3) Overtravel Alarm (OT)

No.	Content	Remarks
1	The movable part of the machine touched the X axis plus stroke limit switch.	
2	The movable part of the machine touched the X axis minus stroke limit switch.	
3	The movable part of the machine touched the Z axis plus stroke limit switch.	
4	The movable part of the machine touched the Z axis minus stroke limit switch.	

(4) Servo Alarm (SV)

No.	Content	Remarks
01	Overload signal of X or Z axis turned on.	
02	The READY signal (VRDY) of velocity control unit has turned off.	
03	The READY signal (VRDY) of velocity control does not turned off even though the READY signal (PRDY) of position control has turned off.	
04	Abnormality of the velocity control system. Normal reference point return cannot performed, that may be generated because of abnormality inside the NC or in servo system. Manual reference point return should be done again.	
11	The content of error register of the X axis is larger than the specified value.	

No.	Content	Remarks
12	The content of error register of the X axis exceeded ±32767. Or the velocity command value of the DA converter is out of the range of –8192 ~ +8101. An incorrect setting will cause this alarm.	
13	A feed rate exceeding 511875 detection unit/sec. was commanded in the X axis. An incorrect setting of CMR causes this alarm.	
14	Feedback signal cable from the X axis servo motor is disconnected.	
15	Drift in X axis is excessive (exceeds 500 VELO)	
21	The content of error register of the Z axis is larger than the specified value.	
22	The content of error register of the Z axis exceeded ±32767. Or the velocity command value of the DA converter is out of the range of –8192 ~ +8191. An incorrect settings will cause this alarm.	
23	A feed rate exceeding 511875 detection unit/sec. was commanded in the Z axis. An incorrect setting of CMR causes this alarm.	
24	Feed back signal cable from the Z axis servo motor is disconnected.	
25	Drift in Z axis is excessive (exceeds 500 VELO)	

(5) Program Error (P/S Alarm)

No.	Content	Remarks
000	Re-apply the power after the parameter was input. (Parameter number 06 ~ 10, 15 ~ 19, 31 ~ 37, 61 ~ 65)	
001	TH alarm (A character with incorrect parity was input.) Correct the tape.	
002	TV alarm (A number of characters in a block is odd.) This alarm will be generated only when the TV check is effective. Correct the tape.	
003	Data exceeding the maximum allowable number of digits was input. (Refer to the item of max. programmable dimensions)	
004	A numeral or the sign (–) was input without an address at the beginning of a block.	
005	The address was not followed by the appropriate data but was followed by another address or EOB code.	
006	Sign (–) input error (Sign (–) was input after an address with which it cannot be used. Or two signs were input.)	
007	Decimal point input error (A decimal point was input after an address with which it cannot be used. Or two decimal points were input.)	
009	Unusable character was input in significant area. (A, B, C, D, E, H, J, L, Q, V, Y)	
010	Unusable G code was commanded.	
011	Feed rate was not commanded at cutting feed or the feed rate was inadequate.	

No.	Content	Remarks
023	In circular interpolation by radius designation, negative value was commanded for address R.	
029	The offset value specified by T code is too large.	
030	The tool offset number is too large for the T function.	
031	In setting of offset amount by G10, the offset number following address P was excessive or it was not specified.	
032	In setting of offset amount by G10, the offset amount was excessive.	
059	The program with the selected number cannot searched, in external program number search.	
060	Commanded sequence number was not found in the sequence number search.	
061	Address P or Q is not specified in G70, G71, G72, or G73 command.	
062	• The cutting depth in G71 or G72 is zero or negative value. • The repetitive count in G73 is zero or negative value. • The negative value is specified to Δi or Δk in G74 or G75. • The zero or negative value is specified to address U or W, though Δi or Δk is not zero in G74 or G75. • The negative value is specified to Δd, though the relief direction in G74 or G75 is determined. • The zero value is specified to the height of thread or cutting depth of 1st time in G76. • The specified minimum cutting depth in G76 is greater than the height of thread. • An unusable angle of tool tip is specified in G76.	
063	The sequence number specified by address P in G70, G71, G72, or G73 command cannot be searched.	
065	• G00 or G01 is not commanded at the block with the sequence number which is specified by address P in G71, G72, or G73 command. • An address Z (W) or X (U) was commanded at the block with sequence number which is specified by address P in G71 or G72, respectively.	
066	An unallowable G code was commanded between two blocks specified address P and Q in G71, G72 or G73.	
067	G70, G71, G72, or G73 command with address P and Q was specified in MDI mode.	
070	The memory area is insufficient.	
071	The address to be searched was not found. Or the program with specified program number was not found in program number search.	
072	The number of programs to be stored exceeded 63.	
073	The commanded program number has already been used.	
074	The program number is other than 1 to 9999.	
076	The address P was not commanded in the block which includes a M98 command.	

No.	Content	Remarks
077	The subprogram was called in triple.	
078	The sequence number which was specified by address P in the block which includes a M98 or M99 was not found.	
079	The contents of the program stored in the memory did not agree with that in tape in collation.	
085	When entering in the memory by using ASR or RS232C interface, an overrun or framing error was generated. The number of bits of input data or setting of baod rate is incorrect.	
086	In entering in the memory by using RS232C interface, the ready signal (DR) of I/O devices was turned off.	
087	When entering data in the memory by using RS232C interface, though the read terminate command is specified, input is not interrupted after 10 characters read.	
090	The reference point return cannot be performed normally because the reference point return start point is too close to the reference point or the speed is too slow.	
092	The commanded axis by G27 (reference point return check) did not return to the reference point.	
100	The switch to set parameters was turned on. Push the reset button after turning off the switch.	
101	The power was turned off while rewriting the contents of the memory in the part program storage & editing operation. When this alarm is generated, you must turn on the power while pushing the DELET and RESET buttons to clear the memory.	

APPENDIX 5 STATUS AT TURNING POWER ON AND AT RESET

○ : The status is not changed or the movement is continued.
× : The status is canceled or the movement is interrupted.

Item		At turning power on	At reset
Setting data	Offset value	○	○
	Setting parameter	○	○
	Parameter	○	○
Data	Program in the memory	○	○
	Sequence number display	×	○
	One-shot G code	×	×
	Modal G code	Initial G code (G20/G21 is not changed)	○
	F function	Zero	○
	S, T or M function	×	○
	Repetitive count	×	×
Coordinate system	Work coordinate value	Zero	○
Movement execution	Movement	×	×
	Dwell	×	×
	Sending of M, S or T function code	×	×
	Tool offset	×	×
	Memorization of called subprogram number	×	× *(Note 1)*
Display LED Output signal	Alarm LED	Extinguishes (if an alarm does not occur)	Extinguishes (if an alarm does not occur)
	READY LED	○	○ (× in emergency stop)
	Reference point return LED	×	○ (× in emergency stop)
	S or T code	×	○
	M code	×	×
	M, S, or T strobe signal	×	×
	Spindle revolution signal (S analog signal)	○	○
	NC ready signal	ON	○
	Servo ready signal	ON (other than servo alarm)	ON (other than servo alarm)
	CYCLE START LED	×	×
	FEED HOLD LED	×	×

(Note 1) When the NC is reset during the subprogram execution, the control return to the start of the main program. A subprogram cannot be executed from the middle of it.

APPENDIX 6 OPERATION TABLE

(1) Operation by Symbolic FAPT function

Classification	Functions	Input data	Input method	Remarks
Processing sequence	Selection of processing sequence	1. Blank & parts (drawing & blank) 2. Blank & parts (part figure) 3. Home position & index position 4. Definition of machining 5. NC data preparation 0. End	Menu selection	Processing can be started with an optional sequence by selecting the sequence using the menu. Processing is started with the first, if no menu is selected.
Blank	Selection of material	1. S45C 2. SCM 3. FC 4. AL 5. SUS 6. Others ⋮	Menu selection	This is used for automatically determining the cutting conditions.
	Selection of coordinate system according to drawing method	1. 1st quadrant 2. 2nd quadrant 3. 3rd quadrant 4. 4th quadrant	Menu selection	The coordinate system and a figure indicating the position of blank are displayed as menu.
	Selection of figure of blank	1. Rod 2. Hollow rod 3. Special figure	Menu selection	For special figures, specify a surplus thickness in the X and Z direction.
	Input of dimension	• Outer diameter • Inner diameter (hollow rod only) • Length • Surplus thickness in X and Z directions (Special figures only)	Key input	
	Coordinate system and blank position	• Distance from X-axis	Key input	

Classification	Functions		Input data	Input method	Remarks
Parts figure	Specification of figure		• Figure element symbol	Symbol key input Kind of figure element symbol → ← ↑ ↓ ↗ ↙ ↖ ↘ ○ ○ T G N C R	The dimension is asked from the system, each time the figure symbol key is depressed according to the figures of cutting drawings. When dimensions are input by the key and a figure is determined, the figure is immediately drawn on the CRT.
	Straight line parallel to X axis [→], [←]	When start point data are available.	• X-coordinate value at start point • Z-coordinate value at start point • X-coordinate value at end point	Key input	Input the dimensions on the machining drawing to the question by the system. When these data are input, the parts figure is determined automatically by the system.
		When start point data are not available.	• X-coordinate value at end point • Z-coordinate value at end point	Key input	
			• Z-coordinate value at end point	Key input	
			• Making tangent to the preceding figure	Key input	
			• Making tangent to the succeeding figure	Key input	
	Straight line parallel to Z axis [↓], [↑]	When start point data are available.	• X-coordinate value at start point • Z-coordinate value at start point • Z-coordinate value at end point	Key input	Input the dimensions on the machining drawing to the question by the system. When these data are input, the parts figure is determined automatically by the system.
		When start point data are not available.	• X-coordinate value at end point • Z-coordinate value at end point	Key input	
			• X-coordinate value at end point	Key input	
			• Making tangent to the preceding figure	Key input	
			• Making tangent to the succeeding figure	Key input	

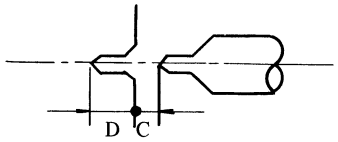
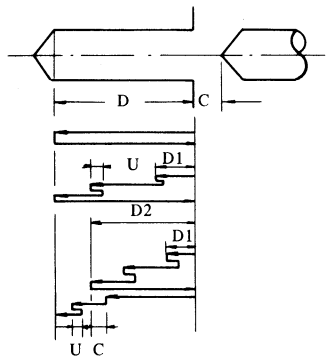
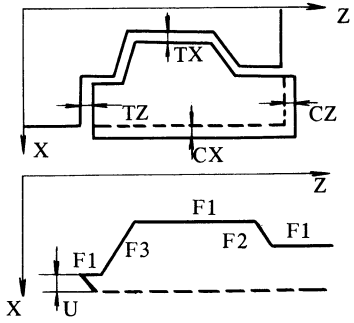
Classification	Function		Input data	Input method	Remarks
Parts figure	Straight line in an optional direction ↖, ↘, ↗, ↙	When start point data are available.	• X-coordinate value at start point • Z-coordinate value at start point • X-coordinate value at end point • Z-coordinate value at end point	Key input	Input the dimensions on the machining drawing to the question by the system. When these data are input, the parts figure is determined automatically by the system.
			• X-coordinate value at start point • Z-coordinate value at start point • Making tangent to the preceding figure	Key input	
			• X-coordinate value at start point • Z-coordinate value at start point • Making tangent to the succeeding figure	Key input	
			• X-coordinate value at start point • Z-coordinate value at start point • Angle with reference to Z-axis	Key input	
		When start point data are not available.	• X-coordinate value at end point • Z-coordinate value at end point • Making tangent to the preceding figure	Key input	Input the dimensions on the machining drawing to the question by the system. When these data are input, the parts figure is determined automatically by the system.
			• X-coordinate value at end point • Z-coordinate value at end point • Making tangent to the succeeding figure	Key input	
			• X-coordinate value at end point • Z-coordinate value at end point • Angle with reference to Z axis	Key input	
			• Making tangent to the preceding figure • Making tangent to the succeeding figure	Key input	

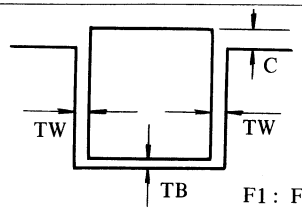
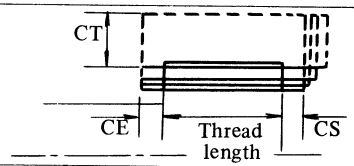
Classification	Function		Input data	Input method	Remarks
Parts figure	Straight line in an optional direction ↖, ↘, ↗, ↙	When start point data are not available.	• Making tangent to the preceding figure • Angle with reference to Z axis	Key input	Input the dimensions on the machining drawing to the question by the system. When these data are input, the parts figure is determined automatically by the system.
			• Making tangent to the succeeding figure • Angle with reference to Z axis	Key input	
	Arc ↻, ↺	When start point data are available.	• X-coordinate value at start point • Z-coordinate value at start point • X-coordinate value at circle center • Z-coordinate value at circle center	Key input	Input the dimensions on the machining drawing to the question by the system. When these data are input, the parts figure is determined automatically by the system.
			• X-coordinate value at start point • Z-coordinate value at start point • X-coordinate value at end point • Z-coordinate value at end point • Radius	Key input	
			• X-coordinate value at start point • Z-coordinate value at start point • Making tangent to the preceding figure • Radius	Key input	
			• X-coordinate value at start point • Z-coordinate value at start point • Making tangent to the succeeding figure • Radius	Key input	
		When start point data are not available.	• X-coordinate value at end point • Z-coordinate value at end point • X-coordinate value at circle center • Z-coordinate value at circle center	Key input	

Classification	Function		Input data	Input method	Remarks
Parts figure	Arc ◠, ◠	When start point data are not available.	• X-coordinate value at end point • Z-coordinate value at end point • Making tangent to the preceding figure • Radius	Key input	Input the dimensions on the machining drawing to the question by the system. When these data are input, the parts figure is determined automatically by the system.
			• X-coordinate value at end point • Z-coordinate value at end point • Making tangent to the succeeding figure • Radius	Key input	
			• X-coordinate value at circle center • Z-coordinate value at circle center • Radius	Key input	
			• X-coordinate value at circle center • Z-coordinate value at circle center • Making tangent to the preceding figure	Key input	
			• X-coordinate value at circle center • Z-coordinate value at circle center • Making tangent to the succeeding figure	Key input	
			• Making tangent to the preceding figure • Making tangent to the succeeding figure • Radius	Key input	
	Threading		• Length • No. of threads • Pitch	Key input	When threading figure symbol [T] is input, input data described on the left are asked from the CRT.
	Grooving		• Width • Depth	Key input	When grooving figure symbol [G] is input, input data described on the left are asked from the CRT.

Classification	Functions		Input data	Input method	Remarks
Parts figure	Chamfering	Straight line parallel to axis	• Size	Key input	When chamfering figure symbol [C] is input, the size is asked from the CRT. Chamfering is applicable to figure elements of all combinations.
		Tapered straight line			
		Arc and straight line			
	Rounding	Straight line parallel to axis	• Radius	Key input	When rounding figure symbol [R] is input, the radius value is asked from the CRT. Rounding is applicable to figure elements of all combinations.
		Tapered straight line			
		Arc and straight line			
	Necking		• Width • Depth	Key input	When necking figure symbol [N] is input, the data for necking as shown left is asked from the CRT.
	Surface roughness		• Numerical value indicating the surface roughness	Key input	When figure symbol is input, the surface roughness is asked from the CRT.
	Four arithmetic operations in data input		• Arithmetic keys [+], [−], [*], [/]	Key input	Four arithmetic operations can be done when data are input, in the same manner as in calculators.
	Function operation in data input		• Function keys [T], [S], [C], [A] [S], [A] [C], [A] [T], [R] and [P]	Key input	Function operations, such as tan, sin, cos, $\sin^{-1}$, $\cos^{-1}$, $\tan^{-1}$, square root and power, can be done when data are input, in the same manner as in calculators.

Classification	Functions	Input data	Input method	Remarks
Machine reference and turret position	Machine reference point	• X-coordinate value • Z-coordinate value	Key input	
	Turret turning position	• X-coordinate value • Z-coordinate value	Key input	
Machining program	Selection of processing contents	1. New 2. Change	Menu selection	
	Process list	• Process No.	Automatic selection of the next process No.	The process to be changed can be selected by shifting the cursor to the desired process No.
	Selection of kind of machining	1. Centering 2. Drilling 3. Rough cutting of outer diameter. 4. Rough cutting of inner diameter 5. Medium finish cutting of outer diameter 6. Medium finish cutting of inner diameter 7. Finish cutting of outer diameter 8. Finish cutting of inner diameter 9. Grooving 10. Threading	Menu selection	
Preparation for machining	Tool number and tool offset number	• Tool number • Tool offset number	Key input	
	Tool figure	• Tool nose radius • Virtual tool position • Cutting edge angle • Tool nose angle or tool width (Grooving tool) • Setting angle • Setting position	Key input (Presettable)	
	Machining start position	• X-coordinate value • Z-coordinate value	Key input (Presettable)	

Classification	Functions	Input data	Input method	Remarks
Preparation for machining	Cutting direction	• ↑ • ↓ • → • ←	Menu selection (Selected by symbolic key)	
	Cutting area	• Start point • End point	Specify the cutting area by the cursor which shifts along the cutting figure	
Cutting conditions	Centering condition	• Clearance quantity (C) • Centering depth (D) • Cutting speed (V) • Feedrate (F)	Key input (Presettable)	
	Drilling conditions	• Clearance quantity (C) • Total depth of drilling (D) • Depth of 1st drilling (D1) • Return amount of 1st drilling (U) • Depth of 2nd drilling (D2) • Clearance of 2nd drilling (C2) • Cutting speed (V) • Feedrate of drilling (F1) • Redrilling feedrate (F2)	Key input (Presstable)	
	Rough cutting conditions	• Clearance quantity (CX) • Clearance quantity (CZ) • Finish allowance (TX) • Finish allowance (TZ) • Cutting depth (D) • Cutting speed (V) • Feedrate (F1) • Feedrate (F2) • Feedrate (F3)	Automatic determination (If a material is already selected.) Key input (If automatically determined value is changed or intermitten input is made) (Presettable)	

Classification	Functions	Input data	Input method	Remarks
Cutting condition	Medium finish and finish cutting conditions	• Clearance quantity (CX) • Clearance quantity (CZ) • Finish allowance (TX) • Finish allowance (TZ) • Cutting speed (V) • Feedrate (F1) • Feedrate (F2) • Feedrate (F3) • Feedrate (F4)	The feedrate is automatically determined, if the surface roughness is already specified. Key input (Presettable)	Only one feedrate applies to the medium finish cutting condition. F1: For roughness 100 μm Rmax F2: For roughness 25 μm Rmax F3: For roughness 6.3 μm Rmax F4: For roughness 0.8 μm Rmax
	Grooving conditions	• Clearance quantity (C) • Finish allowance (TW) • Finish allowance (TB) • Cutting speed (V) • Feedrate (F1) • Feedrate (F2)	Key input (Presettable)	 F1 : For rough cutting F2 : For finish cutting *(Note) None of data TW, TB, F2 is input in case of necking.*
	Threading conditions	• Clearance quantity (CT) • Clearance quantity (CS) • Clearance quantity (CE) • Cutting speed (V)	Key input (Presettable)	
NC data preparation	Program number	• Program number	Key input	A 4-digit program number is outputted.
	NC data preparation			NC data are prepared in the sequence of process program table.
	Drawing of tool path			Blank is drawn to the full size of the CRT. Coordinate axes, blank figure, and tool path, are drawn.

(2) Operation for machining and on NC language

Classification	Function	Key Switch	Parameter Enable switch	Mode Switch	Function display LED	Operation
Clear	Memory All Clear		○	Power ON	–	[PARAM] + [DELET]
	Clearing Parameter		○	Power ON	–	[RESET]
	Clearing Stored Program			Power ON	–	[DELET]
Data Input from Tape	Parameter (Tape → Memory)		○	EDIT	PARAM	[INPUT]
	Offset value			EDIT	OFFSET	[INPUT]
	Program input	○		EDIT/AUTO	PRGRM	[INPUT]
Data Input from MDI	Parameter		○	MDI	PARAM	[SET] → Parameter No. → [INPUT] → Set data → [INPUT] → Parameter Enable Sw. OFF → [RESET]
	Offset value			–	OFSET	[SET] → Offset No. → [INPUT] → Set Offset Value → [INPUT]
	Setting Data			MDI	PARAM	[SET] → Setting No. → [INPUT] → Set data → [INPUT]
Punch out	Parameter			EDIT	PARAM	[START]
	Offset value			EDIT	OFSET	[START]
	All Program			EDIT	PRGRM	O → –9999 → [START]
	One Program			EDIT	PRGRM	O → Program No. → [FWD]
Search	Program No. Search			EDIT/AUTO	PRGRM	O → Program No. → [FWD]
	Sequence No. Search			AUTO	PRGRM	N → Sequence No. → [FWD] (After Program No. Search)
	Address/Word Search			EDIT	PRGRM	Searching data → [FWD]
	Address Search			EDIT	PRGRM	Searching address → [FWD]
Program Editing	Deletion of all Program	○		EDIT	PRGRM	O → –9999 → [DELET]
	Deletion of a Program	○		EDIT	PRGRM	O → Program No. → [DELET]

Classification	Function	Key Switch	Parameter Enable Switch	Mode Switch	Function display LED	Operation
Program Editing	Deletion of Several Blocks	○		EDIT	PRGRM	N → Sequence No. → [DELET]
	Deletion of a Block	○		EDIT	PRGRM	[EOB] → [DELET]
	Deletion of a Word	○		EDIT	PRGRM	Search the word to be deleted → [DELET]
	Alteration of a Word	○		EDIT	PRGRM	Search the word to be altered → [ALTER]
	Insertion of a Word	○		EDIT	PRGRM	Search the word before the place in the program → Set data → [INSRT]
Collation	Collation in Memory with Tape			EDIT/AUTO	PRGRM	[INPUT]
Input to/ output from memory casstte	Program input	○		EDIT/AUTO	PRGRM	N → File No. → [INPUT] [INPUT]
	All programs output			EDIT	PRGRM	O → −9999 → [START]
	Once program output			EDIT	PRGRM	O → Program No. → [START]
	File search			EDIT/AUTO	PRGRM	N → File No., −9999 or −9998 → [INPUT]
	Deletion of file	○		EDIT	PRGRM	N → File No. → [START]
	Collation of program			EDIT/AUTO	PRGRM	N → File No. → [INPUT] [INPUT]

Revision Record

FANUC SYSTEM 3T-MODEL D OPERATOR'S MANUAL (B-53504E)

Edition	Date	Contents	Edition	Date	Contents
02	'82.3	• Addition of Section I.3 (FANUC SYSTEM 3T-MODEL D) • Altaration of Chapter II and III for Symbolic FAPT • Addition of Section IV. 10.17, 10.18, 10.19, 10.21, for input/output operation • Addition of built-in type 2 cabinet. • Change of max. command value for per revolution feed and thread lead. • Addition of parameters "NOFC" (No. 10) and "PSANGN" (No. 36) and deletion of "DRNPR" (No. 12) • Correction of errors.			
01	'81.10	———			

1982.3
(Y1390)
FANUC LTD