PROGRAMMING MANUAL

——FOR

MAZAK

CAM M-2

Type :

SERIAL NUMBER: 55613

Introduction:

The MAZATROL CAM M-2, CNC equipment has been developed for machining centers in which data can be input in daily conversational language. By the use of the CAM M-2 , with its simple & quick programming, a considerable reduction of set-up time can be achieved.

This manual refers mainly to programming of the CAM M-2 . Please read this manual carefully in order to use the CAM M-2 effectively. An operation manual is also available for the customer's convenience.

Caustions:

Since the CAM M-2 has a highly automatic programming function, programming can be completed in a short period of time. However, it is required to perform a tool path check on the graphic display before starting automatic machining since the CAM M-2 cannot process all kinds of machining due to tool path irregularities caused by discontinuity of calculated machining configuration.

Please note that no other function than mentioned in the manual provided.

The specifications of the machine are subject to change for improvement without advance notice.

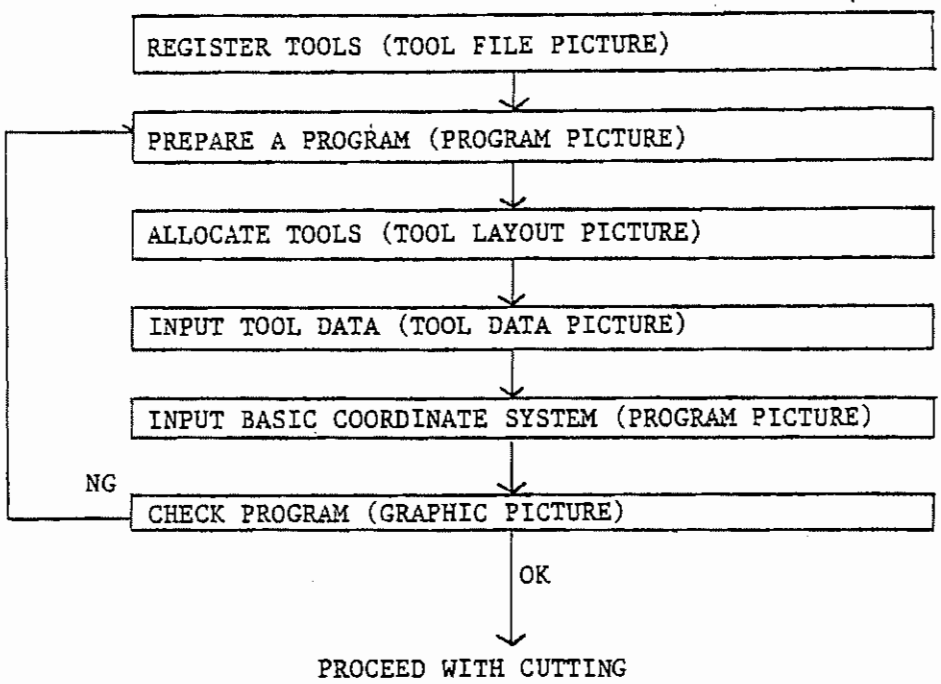
CONTENTS

1. OVERVIEW

A program is prepared to communicate with the NC unit using a conversational language (MAZATROL Language).

Corresponding to a position of the cursor, a message and a menu will be displayed in the lower part of a picture. According to the message, therefore, input data using a menu key or the ten keys.

Before proceeding to cut with CAM M-2 , prepare a program and input a variety of data, following the procedure given below.



This chapter describes MAZATROL program preparation only. For other items, see the operating manual.

2. FUNCTIONS OF AUTOMATIC PROGRAMMING

2.1 Program and Unit Composition

■ Composition of Program

In principle, the program for one workpiece consists of the commom unit data, fundamental coordinate system unit data, machining unit data(tool sequence data and shape sequence data) and end unit data(and auxiliary coordinate system unit data and subprogram unit data etc., if necessary)

(1) Common unit data

Only one common unit data can be set at the head of the program relating to data on material to be machined, etc.

(2) Fundamental coordinate system unit data

Data on relative position in the machine coordinates and workpiece coordinates.

(3) Machining unit data

Data on selection of machining method, etc.

(i) Tool sequence data

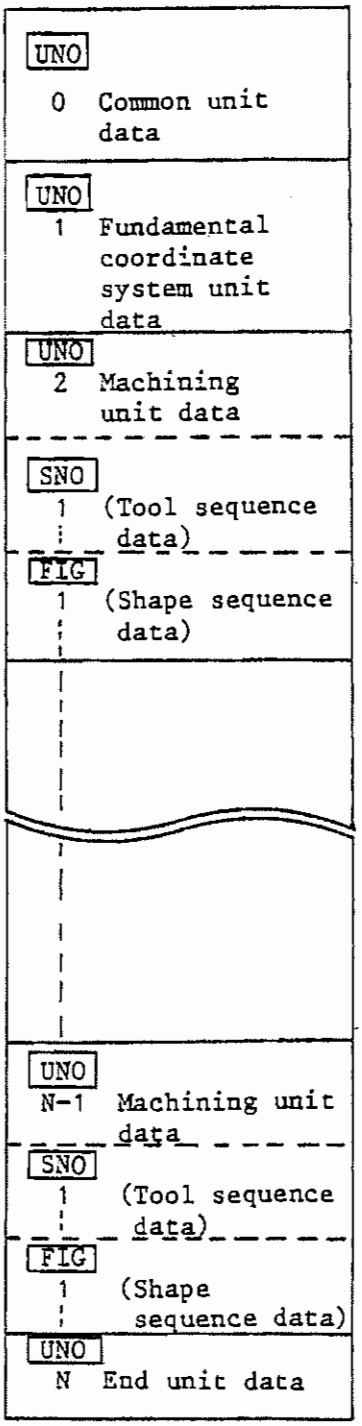
Data on tool operation mode and associated with it

(ii) Shape sequence data

Data associated with machining dimensions on drawing

(4) End unit data

Data on procedures practiced upon completion of machining



■ Composition of Unit

To make the program simpler, kinds of machining are set in the "unit" basis. Each unit is further divided into the tool sequence and shape sequence.

Unit	Purpose	Sequence
COMMON UNIT	It is always displayed at the head of each program to set the kind of material used, etc.	None
POINT MACH-ING	Used to select hole machining	Tool sequence and shape sequence
LINE MACH-ING	Used to select line machining	Tool sequence and shape sequence
FACE MACH-ING	Used to select surface machining	Tool sequence and shape sequence
MANUAL PROGRAM	Used to execute program, utilizing G code, M code, etc.	Shape sequence
OTHER	Call sub program Used to execute program containing many M codes Input a drum number (DRUM CHG) Automatically measure basic coordinates (MMS) Setting the pallet change No. (PALL CHG) Control of machined surface angle (Index) End of priority given to the same kinds tools (PROCESS END)	None (Shape sequence is available in the MMS unit only.)
WPC	Used to set fundamental coordinate system (WPC)	None
OFFSET	Used to set auxiliary coordinate system (OFFSET)	None
END	Used to set end	None

2.1.1 Common unit

The following unit is always displayed at the head of a program which is going to be developed.

UNO	MAT	INITIAL-Z	MULTI MODE	MULTI	PITCH-X	PITCH-Y
0						

MAT: Material of workpiece is inputted with the menu key.

CAST IRN	DUCTILE CAST IRN	CARBON STEEL	ALLOY STEEL	STAINLES STEEL	ALMINIUM	COPPER ALLOY		OTHER

CAST IRN ... Gray cast iron

DUCTILE CAST IRN ... Spheroidal graphite cast iron

CARBON STEEL ... Carbon steel for machinery structures

ALLOY STEEL ... Chromium-molybdenum steel

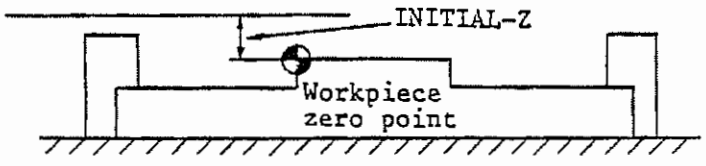
STAINLESS STEEL... Stainless steel

ALMINIUM ... Aluminium alloy

COPPER ALLOY ... Copper alloy

OTHER ... 1 through 8 can be set. (The parameter used for automatic selection of the cutting condition is put in the first screen for parameters "CUT COND. PARAM".

INITIAL-Z: This defines the Z plane (*) where the tool does not have its cutting edge interfere with any workpiece, jig or the like even if it moves X- and Y- axially. It is commanded in coordinates (Z=) from the zero point of the workpiece. This value does not vary even when the auxiliary coordinate system is used.



* In auto mode operation, this height is used for positioning the tool.

MULTI MODE: Kind of multi-piece machining is selected by menu key.

MULTI OFF	MULTI 5*2	OFFSET TYPE						

MULTI: Select only when the menu key [MULTI 5*2] is depressed.

PITCH-X: Pitch in machining a number of workpieces in the X direction.

PITCH-Y: Pitch in machining a number of workpieces in the Y direction.

Note: In the multi-piece machining program, the graphic running order for the tool path on the graphic display differs from those for the shape. The tool path is checked in accordance with the actual cutting.

About Multi-Piece Machining:

(1) MULTI OFF: Multi-piece machining is not executed.

(2) MULTI 5*2:

MULTI MODE MULTI PITCH-X PITCH-Y

5*2

10	9	8	7	6	5	4	3	2	1
0	0	0	0	0	0	0	0	0	0

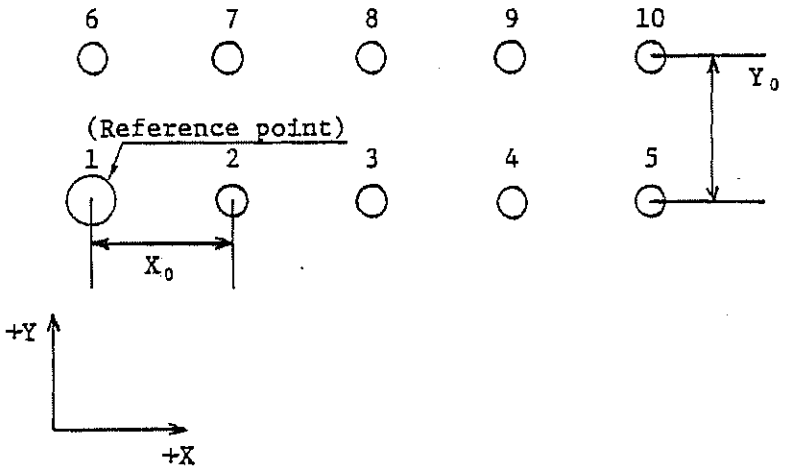
Xo Yo

(Either "1" (executed) or "0" (not executed) is set.)

A maximum of ten workpieces can be machined at a time.

Among 1 through 10, the lower five digits indicate that the workpieces in the lower row should be machined (or not machined). The upper five digits are for the upper row. "1" indicates the position of the workpiece used as reference point. The other workpieces are to be placed at the pitches of X and Y.

Relative positions of workpieces



Note 1:Multi-workpiece machining in the process with the manual program unit requires absolute 3 axes to be commanded in the initial sequence.

Note 2:The M code entered at the end of a tool sequence is outputted each time machining is performed at each multi-workpiece point.
When selected in the M code unit, the M code will be outputted only once.

(3) OFFSET TYPE

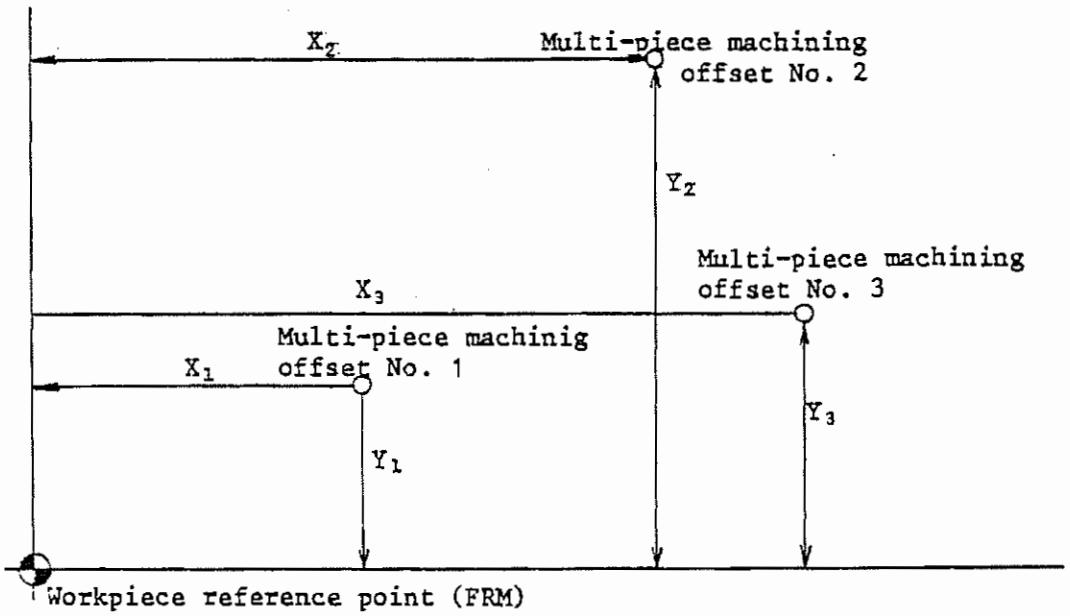
If the offset is effected in the multi-piece machining mode, a maximum of ten data can be entered from the zero point in the programmed fundamental coordinates as offset data for common units.

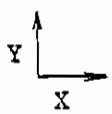
Depressing the [OFFSET END] menu key prepares the system to receive the next unit data input.

							OFFSET END	

[UNO]	MAT	INITIAL-Z	MULTI MODE	MULTI	PITCH-X	PITCH-Y
0			OFFSET TYPE	◆	◆	◆
[OFS]		X	Y		Θ	Z
1		X_1	Y_1		Θ_1	Z_1
2		X_2	Y_2		Θ_2	Z_2
3		X_3	Y_3		Θ_3	Z_3
↓		↓	↓		↓	↓

Relative workpiece positions





Machining is executed in the coordinate system defined by offset data.

Multi-piece machining nesting function

In the multi-piece machining mode, a number of workpiece are machined at a time. This function may be nested in a subprogram to allow complexed pattern machining.

A - D : Unit number

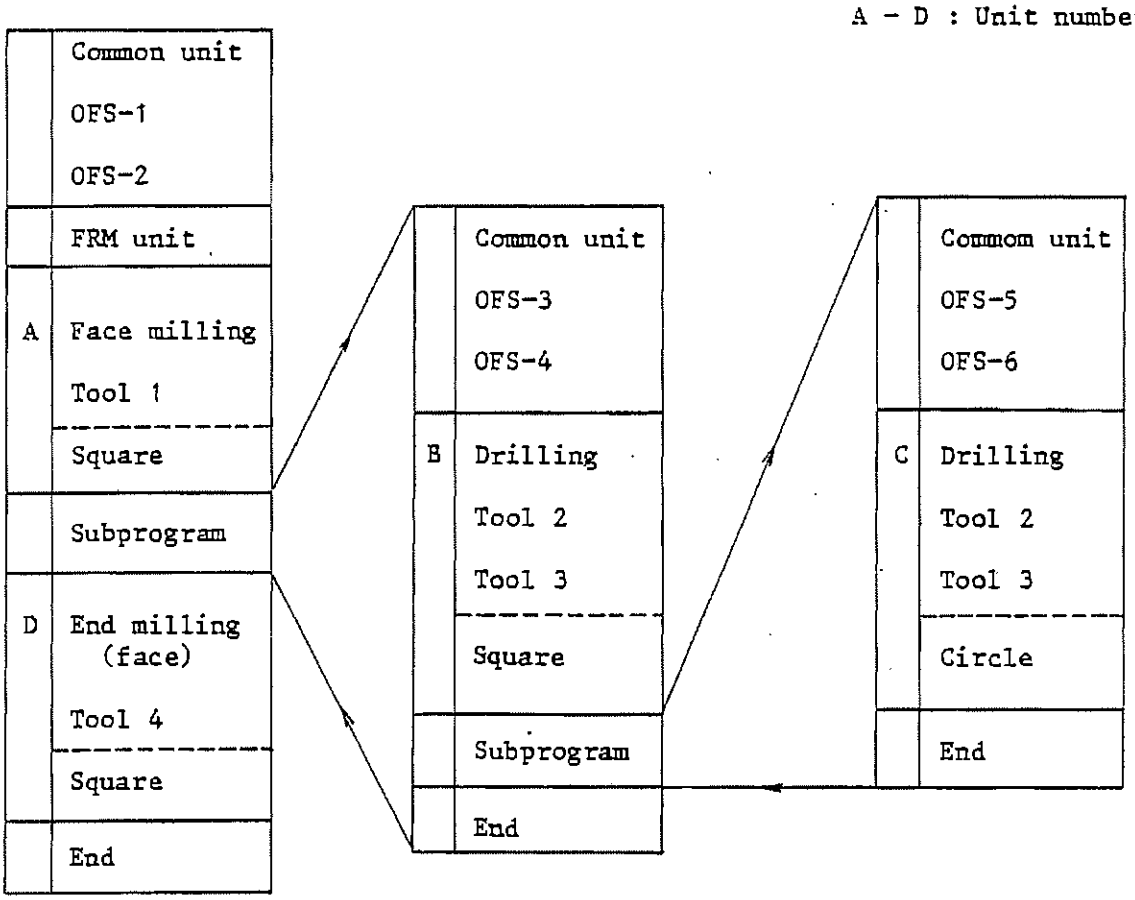


Fig. 1

The machining patternes shown in Fig. 1 may be expressed
is shown in Fig. 2
Each offset is nested against the workpiece zero point (FRM).

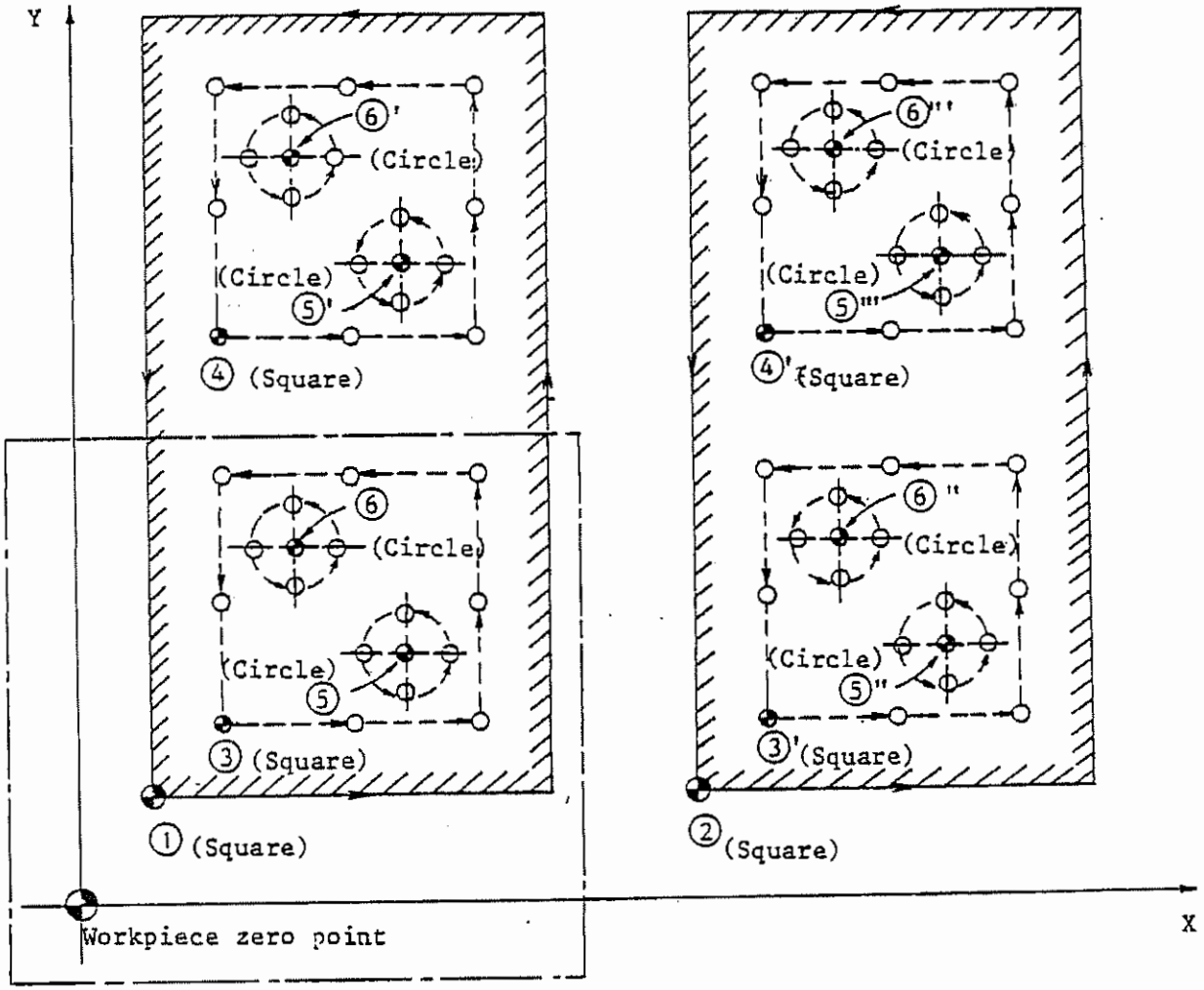
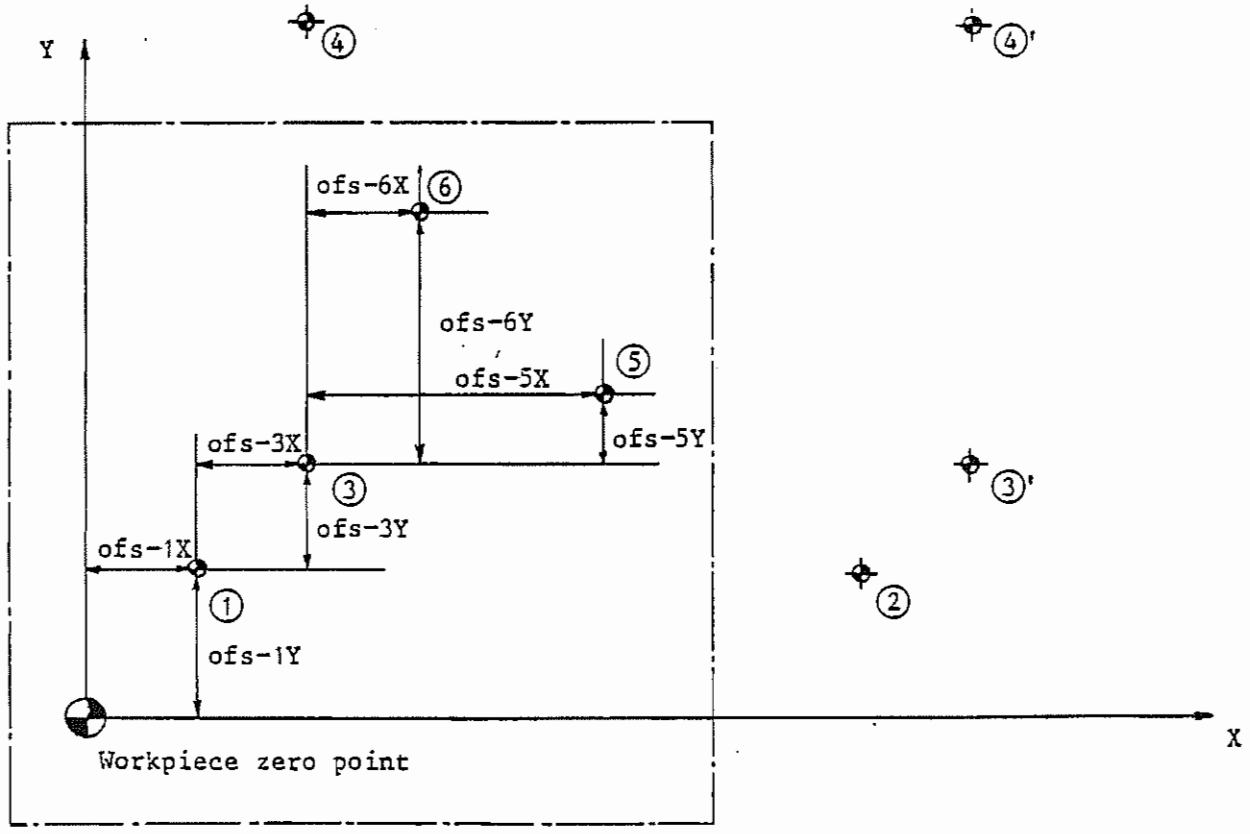


Fig. 2

The respective amount of offset, ① through ⑥ ''' , in each machining shown in Fig. 2 will be as follows:

①	ofs-1	②	ofs-2
③	(ofs-1) + (ofs-3)	③'	(ofs-2) + (ofs-3)
④	(ofs-1) + (ofs-4)	④'	(ofs-2) + (ofs-4)
⑤	(ofs-1) + (ofs-3) + (ofs-5)	⑤''	(ofs-2) + (ofs-3) + (ofs-5)
⑥	(ofs-1) + (ofs-3) + (ofs-6)	⑥''	(ofs-2) + (ofs-3) + (ofs-6)
⑤'	(ofs-1) + (ofs-4) + (ofs-5)	⑤'''	(ofs-2) + (ofs-4) + (ofs-5)
⑥'	(ofs-1) + (ofs-4) + (ofs-6)	⑥'''	(ofs-2) + (ofs-4) + (ofs-6)

Next, how the offset is set will be detailed citing the section surrounded by the broken lines in Fig. 2 as an example.
(Construe that this will also apply to the offset in other sections.)



Order of multi-piece machining nesting

In multi-piece machining, offset starts at the block having the greatest loop. When the machining enters into a subprogram and encounters multi-piece machining offset, it is added to the offset at the outside. The deepest portion is machined as frequently as the product obtained by multiplication of the frequencise of multi-piece machining in each level. Therefore, the order of machining in Fig. 1 is as follows:

Table 1

Order	Offset amount(No. of Fig. 2)		Unit No.	Tool No.
1	ofs-1	①	A	1
2	ofs-2	②	A	1
3	(ofs-1) + (ofs-3)	③	B	2
4	(ofs-1) + (ofs-4)	④	B	2
5	(ofs-2) + (ofs-3)	③'	B	2
6	(ofs-2) + (ofs-4)	④'	B	2
7	(ofs-1) + (ofs-3)	③	B	3
8	(ofs-1) + (ofs-4)	④	B	3
9	(ofs-2) + (ofs-3)	③'	B	3
10	(ofs-2) + (ofs-4)	④'	B	3
11	(ofs-1) + (ofs-3) + (ofs-5)	⑤	C	2
12	(ofs-1) + (ofs-3) + (ofs-6)	⑥	C	2
13	(ofs-1) + (ofs-4) + (ofs-5)	⑤'	C	2
14	(ofs-1) + (ofs-4) + (ofs-6)	⑥'	C	2
15	(ofs-2) + (ofs-3) + (ofs-5)	⑤''	C	2
16	(ofs-2) + (ofs-3) + (ofs-6)	⑥''	C	2
17	(ofs-2) + (ofs-4) + (ofs-5)	⑤'''	C	2
18	(ofs-2) + (ofs-4) + (ofs-6)	⑥'''	C	2
19	(ofs-1) + (ofs-3) + (ofs-5)	⑤	C	3
20	(ofs-1) + (ofs-3) + (ofs-6)	⑥	C	3
21	(ofs-1) + (ofs-4) + (ofs-5)	⑤'	C	3
22	(ofs-1) + (ofs-4) + (ofs-6)	⑥'	C	3
23	(ofs-2) + (ofs-3) + (ofs-5)	⑤''	C	3
24	(ofs-2) + (ofs-3) + (ofs-6)	⑥''	C	3
25	(ofs-2) + (ofs-4) + (ofs-5)	⑤'''	C	3
26	(ofs-2) + (ofs-4) + (ofs-6)	⑥'''	C	3
27	ofs-1	①	D	4
28	ofs-2	②	D	4

2.1.2 Fundamental coordinate system unit

Depress the [WPC] mode menu key.

POINT MACH-ING	LINE MACH-ING	FACE MACH-ING	MANUAL PROGRAM	OTHER	WPC	OFFSET	END	GROUP CHECK

The following display will appear on the screen:

UNO	UNIT	X	Y	θ	Z	4
	WPC-0 (M)					

① It is necessary to tell the NC unit at which position of the machine the workpiece has been installed. This unit sets the distance between the original point position and the original machine point in the work program. This unit is set after the common unit.

② Contents of data (See the figure on the next page.)

WPC- : Many WPC's can be set in one program. A figure is set here for identification. (0 ∿ 99)

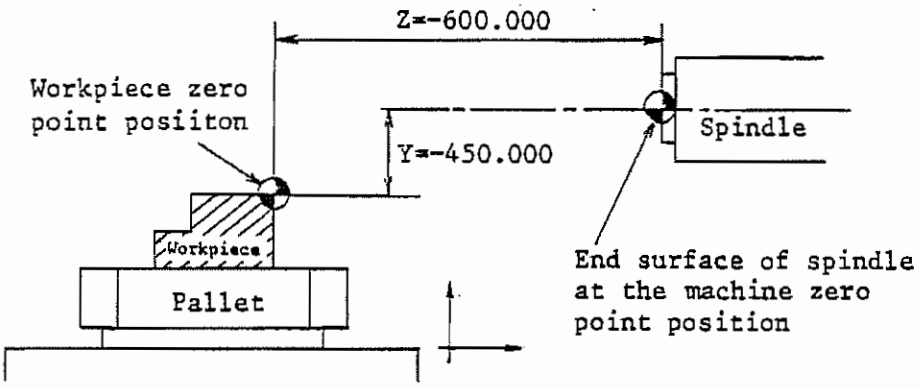
X,Y,Z,4 : Coordinates of the original workpiece position as seen from the zero machine position. (0 to ±9999.999)

θ : Angle formed by the machine coordinates and work coordinates. (0 to ±999.999)

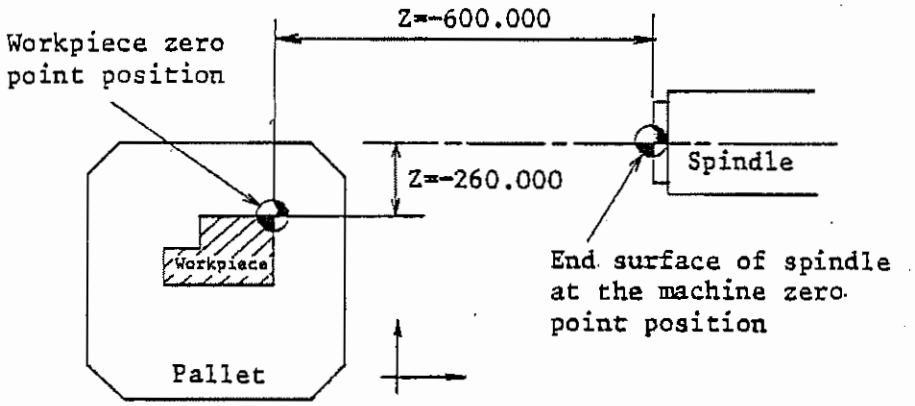
③ The measured values are entered as X, Y, Z, 4 and θ. During programming, these columns may be left blank and measured values may be entered just before starting of machining.

UNO	UNIT	X	Y	θ	Z	4
1	WPC-0	-260.000	-450.000	θ_0	-600.000	0

(Y-Z axis)

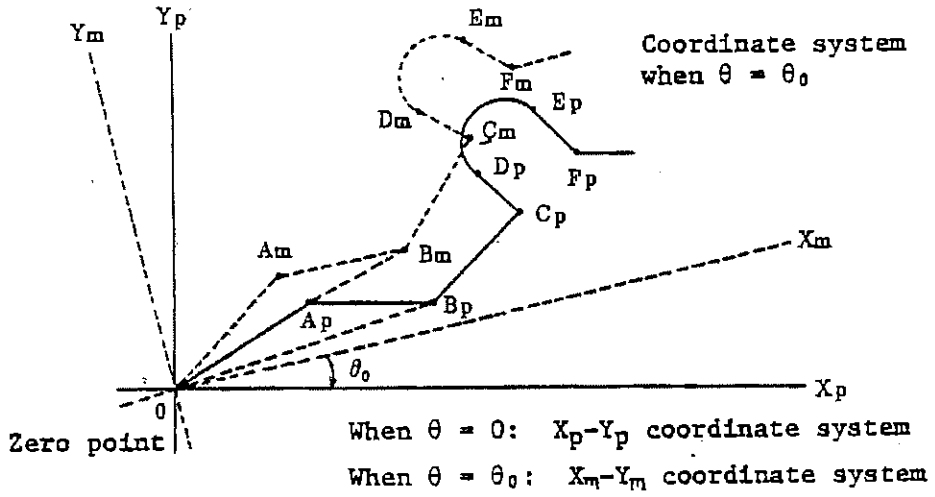


(X-Z axis)



Note: This illustration represents a horizontal type machining center.

④ If the reference line of the material deviates from the machine coordinate system upon installation of the material to be machined, the machine can be controlled by rotating the coordinate system of the machining program. Data is set to θ. Then, θ forms the coordinate system as shown below:



Program coordinate locus (θ = 0)

Ap - Bp - Cp - Dp - Ep - Fp -

Machine coordinate locus (θ = θ_0)

Am - Bm - Cm - Dm - Em - Fm -

Formula for conversion from program coordinate system position to machine coordinate system position

Program coordinate system position Np (Xap, Yap) a = A, B, C ...

Machine coordinate system position Nm (Xam, Yam)

$Kam = Xap \cdot \cos\theta_0 - Yap \cdot \sin\theta_0$

$Yam = Xap \cdot \sin\theta_0 - Yap \cdot \cos\theta_0$

2.1.3 Point(hole) machining mode unit

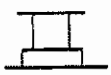
Depress the mode menu [POINT MACH-ING].

POINT MACH-ING	LINE MACH-ING	FACE MACH-ING	MANUAL PROGRAM	OTHER	WPC	OFFSET	END	GROUP CHECK

Based on the shape of a hole to be machined, a machining unit can be selected in a menu.

(1) Unit data

Machining unit selection menu (I)

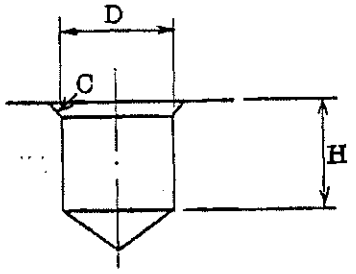
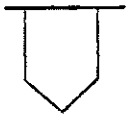
DRILLING	RGH CBOR	RGH BCB	REAMING	TAPPING	BORING	BK-CBORE	CIRC MIL	CBOR-TAP
								

Machining unit selection menu(II) [BORING].
Select a boring unit out of the following four:

BORING T1	BORING S1	BORING T2	BORING S2					

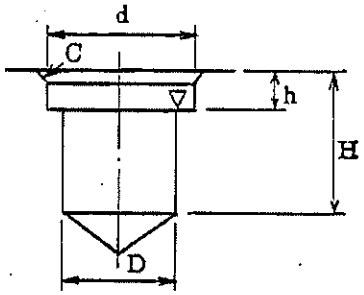
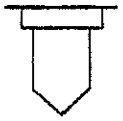
Note: indications parenthesized will not actually appear on the picture (M) represents an input with a menu key.

POINT/MACH-ING mode machining unit

Unit Name	DRILLING	
Menu Key	DRILLING 	

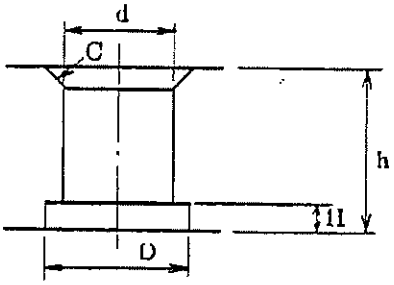
UNIT	DIA(D)	DEPTH(H)	CHMF(C)
DRILL	999.999	999.999	C99.9
(M)			

Tools automatically operated: Spot drill, drill, chamfering cutter

Unit Name	RGH CBOR Machining	
Menu Key	RGH CBOR 	

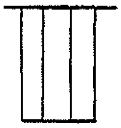
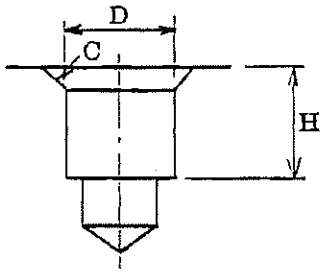
UNIT	CB-DIA(d)	CB-DEP(h)	CHMF(c)	BTM(▽)	DIA(D)	DEPTH(H)
RGH CBOR	999.999	999.999	C99.9	9	999.999	999.999
(M)				(M) (Note 1)		

Tools automatically operated: Spot drill, drill, end mill and chamfering cutter

Unit Name	RGH BCB Machining	
Menu Key	RGH BCB 	

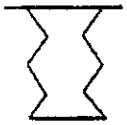
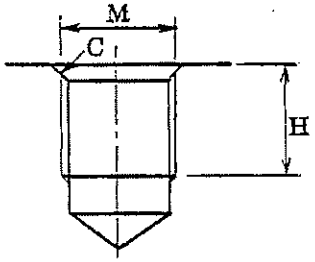
UNIT	CB-DIA(D)	CB-DEP(H)	DIA(d)	DEPTH(h)	CHMF(C)
RGH BCB	999.999	999.999	999.999	999.999	C99.9
(M)					

Tools automatically operated: Spot drill, drill and back facing tool.

Unit Name	REAMING	
Menu Key	REAMING 	

UNIT	DIA(D)	DEPTH(H)	CHMF(C)	PRE-REAM	CHP
REAM (M)	999.999	999.999	C99.9	END MIL (M) (Note 2)	1

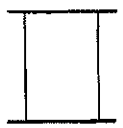
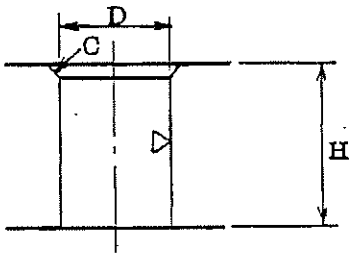
Tool automatically operated: Spot drill, drill, end mill (drill, boring) chamfering cutter, chip absorber and reamer

Unit Name	TAPPING	
Menu Key	TAPPING 	

UNIT	NOM-	MAJOR-ϕ(M)	PITCH(P)	TAP-DEP(H)	CHMF(C)	CHP
TAP (M)	No.12-24UN (M) (Note3)	999.999 (AUTO)	P9.999 (AUTO)	999.999	C99.9 (AUTO)	1

Tool automatically operated: Spot drill, drill, chamfering cutter, chip absorber and tap

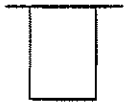
BORING HOLE Unit - (a)

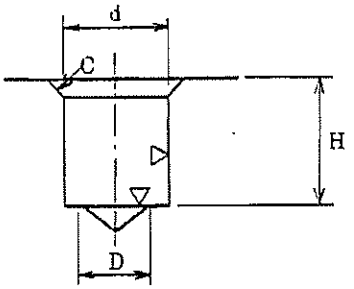
Unit Name	Pierced hole boring BORE T	
Menu Key	BORING  T_1	

UNIT	DIA(D)	DEPTH(H)	CHMF(C)	WAL[illegible](▽)
BORE T_1 (M)	999.999	999.999	C99.9	9 (M)

Tool automatically operated: Spot drill, drill, end mill, boring and chamfering cutter

BORING HOLE Unit - (b)

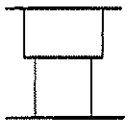
Unit Name	Unpierced hole boring BORE S_1
Menu Key	BORING  S_1

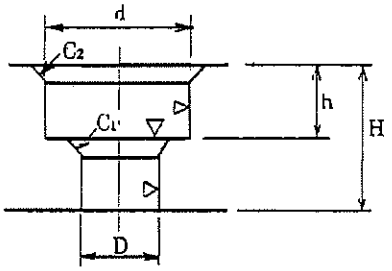


UNIT	DIA(d)	DEPTH(H)	CHMF(C)	BTM(▽)	WAL■■(▽)	PRE-DIA(D)
BORE S_1 (M)	999.999	999.999	C99.9	9 (M)	9 (M)	999.999

Tool automatically operated: Spot drill, drill, end mill, boring and chamfering cutter

BORING HOLE Unit - (c)

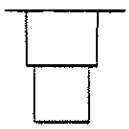
Unit Name	Pierced stepped hole boring BORE T_2
Menu Key	BORING  T_2

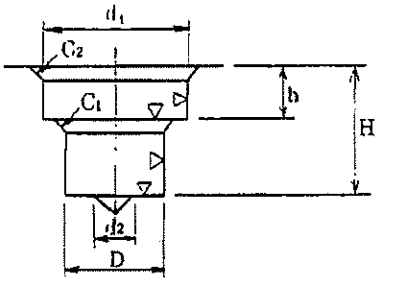


UNIT	CB-DIA(d)	CB-DEP(h)	CHMF(C_2)	BTM(▽)	WAL■■(▽)	DIA(D)	DEPTH(H)	CHMF(C_1)	WAL■■(▽)
BORE T_2 (M)	999.999	999.999	C99.9	9 (M)	9 (M)	999.999	999.999	C99.9	9 (M)

Tool automatically operated: Spot drill, drill, end mill, boring and chamfering cutter

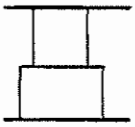
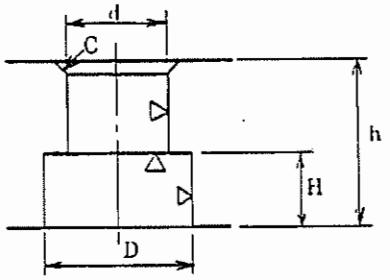
BORING HOLE Unit - (d)

Unit Name	Unpierced stepped boring BORE S_2
Menu Key	BORING  S_2



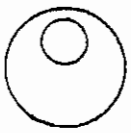
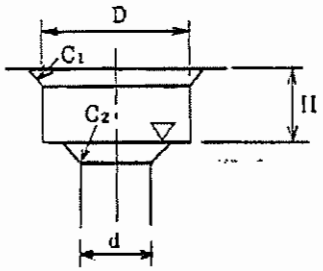
UNIT	CB-DIA(d_1)	CB-DEP(h)	CHMF(C_2)	BTM(V)	WAL■■(V)	PRE-DIA (d2)	DIA(D)	DEPTH(H)	CHMF(C_1)	BTM(▽)	WAL■■(▽)
BORE S_2 (M)	999.999	999.999	C99.9	9 (M)	9 (M)	999.999	999.999	999.999	C99.9	9 (M)	9 (M)

Tool automatically operated: Spot drill, drill, end mill, boring and chamfering cutter

Unit Name	Back boring	
Menu Key	BK-CBORE 	

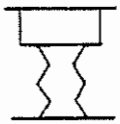
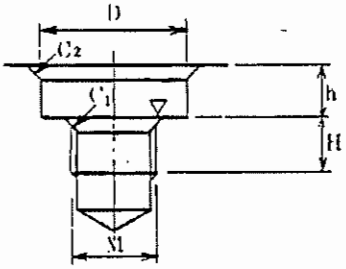
UNIT	DIA(D)	DEPTH(H)	BTM(▽)	WAL[illegible](C)	PRE-DIA(d)	PRE-DEP(h)	CHMF(C)	WAL[illegible](▽)
BK-CBORE (M)	999.999	999.999	9 (M)	9 (M)	999.999	999.999	C99.9	9 (M)

Tool automatically operated: Spot drill, drill, end mill, boring, chamfering cutter, back facing bore

Unit Name	Circle milling	
Menu Key	CIRC MIL 	

UNIT	DIA(D)	DEPTH(H)	CHMF(C)	BTM(▽)	PRE-DIA(d)	CHMF(C2)
CIRC MIL (M)	999.999	999.999	C99.9	9 (M)	999.999	C99.9

Tools automatically operated: End mill and chamfering cutter

Unit Name	CBOR tapping	
Menu Key	CBOR-TAP 	

UNIT	NOM-	MAJOR-Ø(M)	PITCH(P)	TAP-DEP(H)	CHMF(C1)	CB-DIA(D)	CB-DEP(h)	CHMF(C2)	BTM(V)	CHP
CBOR TAP (M)	No.12-28UN (M) (Note 1)	999.999 (AUTO)	P9.999 (AUTO)	999.999	C99.9 (AUTO)	999.999	999.999	C99.9	9 (M)	1

Tools automatically operated: Spot drill, drill, end mill, chamfering cutter, chip collector and tap.

Menus available when machining unit shape definition data are entered:

(Note 1) With cursor ⟶ BTM

▽ 1	▽ 2	▽▽ 3	▽▽ 4	▽▽▽ 5	▽▽▽ 6	▽▽▽ 7	▽▽▽▽ 8	▽▽▽▽ 9

(Note 2) With cursor ⟶ PRE-REAM (for reamer cycle only)

DRILLING	BORING	END MILLING						

(Note 3) With cursor ⟶ NOM- (for tap and seated tap cycle only)

METRIC THRD(M)	UNFY THRD(UN)	PIPE THRD(PT)	PIPE THRD(PF)	PIPE THRD(PS)				OTHER

When UNFY THRD is depressed, with cursor⟶ NOM- (for tap and seated tap cycle only)

NO	H(1/2) HALF	Q(1/4) QUARTER	E(1/8) EIGHTH	S(1/16) SIXTENTH				

When PIPE THRD is deprassed, with cursor⟶ NOM-(for tap and seated tap cycle only)

	H(1/2) HALF	Q(1/4) QUARTER	E(1/8) EIGHTH					

For the procedures of input, refer to " Point Machining (TAP) of (v) Practical Examples by Unit in 3.1 Procedure ".

(2) Tool sequence data

Tools are automatically operated (developed) according to the machining unit.

☐ Kinds of tools:

Spot drill (CTR-DR)
Drill (DRILL)
Chamfering cutter (CHF-M)
End mill (E-MILL)
Back facing tool (BK FACE)
Reamer (REAM)
Tap (TAP)
Boring tool (BOR BAR)
Back boring tool (B-B BAR)
Chip collector (CHP VAC)

☐ Contents of sequence:

◆ ... unnecessary to set

	TOOL	MOM-φ	NO.	HOLE-φ	HOLE DEP	PRE-DIA	PRE-DEP	RGH	DEPTH	C-SP	FR	M	M
CTR-DR	①	②	③	④	◆	◆	◆	(Note 8a)	◆	⑩	⑪	⑫	⑫
DRILL	①	②	③	④	⑤	◆	◆	(Note 8b)	(Note 9a)	⑩	⑪	⑫	⑫
CHF-M	①	(Note 2)	③	(Note 4)	(Note 5)	⑥	⑦	◆	(Note 9b)	⑩	⑪	⑫	⑫
E-MILL	①	(Note 2)	③	④	⑤	⑥	◆	⑧	(Note 9c)	⑩	⑪	⑫	⑫
BK FACE	①	②	③	④	⑤	◆	(Note 7a)	◆	◆	⑩	⑪	⑫	⑫
REAM	①	②	③	④	⑤	◆	◆	◆	(Note 9d)	⑩	⑪	⑫	⑫
TAP	①	②	③	④	⑤	◆	◆	(Note 8c)	(Note 9e)	⑩	⑪	⑫	⑫
BOR BAR	①	②	③	④	⑤	(Note 6a)	(Note 7b)	⑧	(Note 9f)	⑩	⑪	⑫	⑫
B-B BAR	①	②	③	④	⑤	(Note 6b)	(Note 7a)	⑧	(Note 9f)	⑩	⑪	⑫	⑫
CHP VAC	(Note 1)	②	③	◆	◆	◆	◆	◆	◆	⑩	⑪	⑫	⑫

When the machining unit shape data have been entered, tools used to machine a workpiece according to the machining unit shape are automatically developed in the order of machining. Then, items shown in the list are displayed for each tool.

① Tool name (automatically selected) [TOOL]

The tool may be exchanged with the use of the menu keys.

CENTER DRILL	DRILL	CHMF. CUTTER	ENDMILL	BACKSPOT FACER	REAMER	TAP	BORING BAR	BACK BOR.BAR

This is used for editing only.
(Normally the tool determined automatically)

Note 1: The chip collector is not on the menu. It can only be selected automatically.

② Nominal tool diameter [NOM-Ø]

The end mill (E-MILL) and chamfering cutter (CHF-M) must be set with the numerical keys. Other tools are automatically set. The tools of the same kind are discriminated with the menu keys.

WITHOUT ID CODE	A	B	C	D	E	F	G	H

The nominal diameter is automatically displayed. In case tools have different length or is made of different materials, they are descriminated by adding suffix to them.

Note 2 : Alarm will result, if the end mill and chamfering cutter has not been registered in the tool file beforehand.

③ Tool priority machining Nos. [NO.]

Proceed to input of tool priority machining numbers (prior machining No./subsequent machining No.).
For input of machining numbers, menu keys and/or ten keys are used:

(1) Prior machining No.
It is input with ten keys only. (1-63)

(2) Subsequent machining No.
The DELAY PRIORITY menu key is depressed to inverse the menu. Then, the subsequent machining No. is entered with ten keys. (1-63)

(3) None (Input is unnecessary.)

Menu

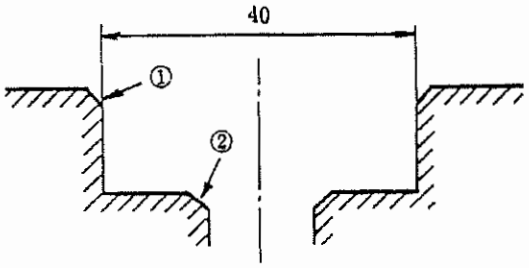
	DELAY PRIORITY		PRI. NO. CHANGE	PRI. NO ASSIGN		PRI. NO ALL ERAS	SUB PRO= PROC END	

For details, see "2.2 Same Tool Priority Machining Function".

④ Machined hole diameter (automatically set) [HOLE-Ø]

Note 4: For a chamfering cutter, this item equals twice the distance from its center to the edge. If there is no interference, 999 is set. It means "D_0" in the figure below.

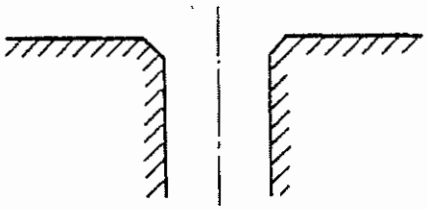
Example 1: Facing a seated hole



① ... D_0 = 999

② ... D_0 = 40

Example 2: Other hole

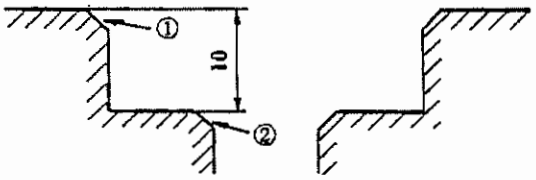


D_0 = 999

⑤ Depth of machined hole (automatically set) [HOLE DEP]

Note 5: This item means H_0 in the figure below for a chamfering cutter.

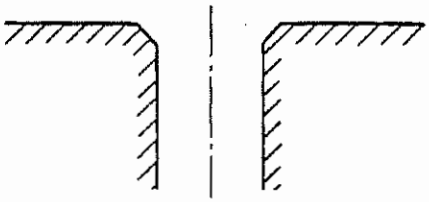
Example 1: Chamfering a seated hole



① ... H_0 = 0

② ... H_0 = 10

Example 2: Other hole



H_0 = 0

⑥ Lower hole diameter (automatically set) [PRE-DIA]

Note 6a: With boring, the boring cycle is entered in this item. Select a boring cycle from the menu and input it. During development of tools, "CYCLE 1" is displayed in this item unconditionally.

CYCLE 1	CYCLE 2	CYCLE 3						

CYCLE 1: Fine boring cycle

CYCLE 2: Rough boring cycle (returned from hole bottom at quick feed speed)

CYCLE 3: Rough boring cycle (returned from hole bottom at a rate set by the parameter EM1)

Note 6b For a back boring tool, this item means the through hole diameter.

⑦ Lower hole depth (automatically set) [PRE-DEP]

Note 7a: In back facing and back boring, this item means the depth of the through hole.

Note 7b: In boring, this item means depth of the seat. Therefore, "0" is set for the through hole bore or stop-end hole bore.

⑧ Cut surface roughness [RGH]

Set with menu keys

▽	▽	▽▽	▽▽	▽▽▽	▽▽▽	▽▽▽	▽▽▽▽	▽▽▽▽
1	2	3	4	5	6	7	8	9

Note 8a: When spot drilling is to be executed, the cutting edge angle is selected.

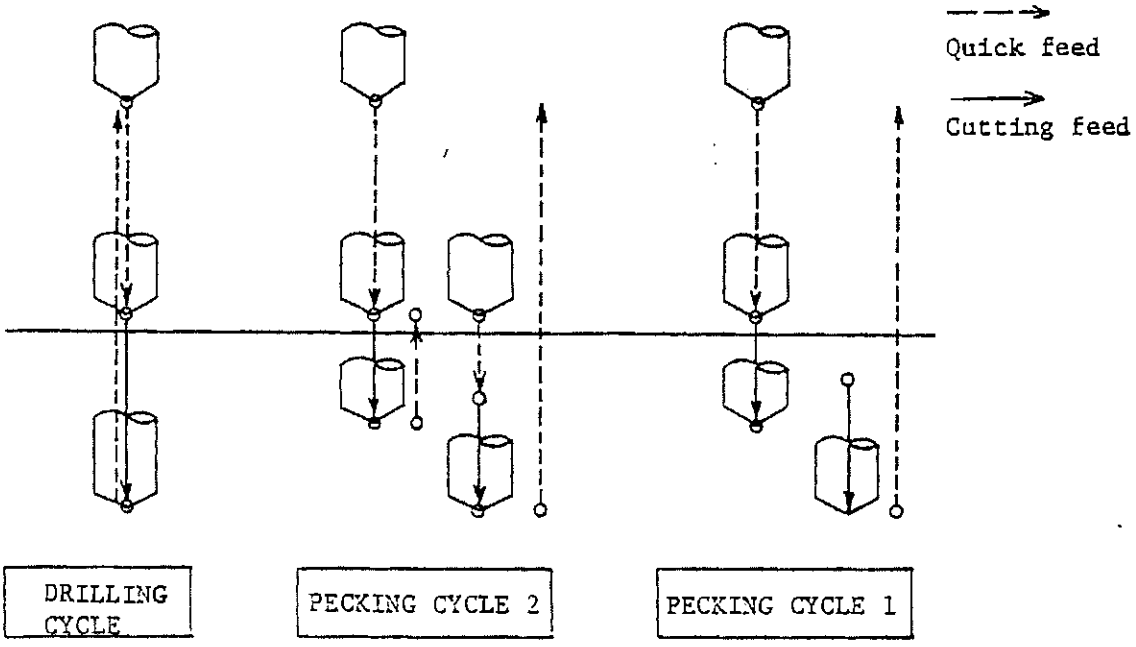
During development of tools, "90°" is displayed in this item by itself.

90°	118°							

Note 8b: For drills, the drill cycle must be set with menu keys.

DRILLING CYCLE	PECKING CYCLE 1	PECKING CYCLE 2						

This defines the drilling cycle. Generally, it is determined automatically from the machining depth and drill diameter. For movement of tools, see the figure below.



Note 8c: For tapping, the doweling time is entered.

Using ten keys, input the doweling time.
During development of tools, "FIX" is displayed in this item by itself.

⑨ Penetration (automatically determined) [DEPTH]

Note 9a: For drills, it means the amount of penetration achieved by one operation.

Note 9b: Amount of chamfering for chamfering tools

Note 9c: For end mills, this item means the amount of penetration in the direction of radius per operation.

Note 9d: For reaming the reamer return speed is entered.
Input data is set with menu keys or ten keys;
During development of tools, "G01" is displayed in this item by itself.

JOG G01	RAPID G00							

When "G01" is selected, the reamer will return at the speed set by the EM1 parameter.

Note 9e: Pitch of thread in case of tap.

Note 9f: Amount of penetration in radius direction in case of boring and back boring.

⑩ Peripheral speed [C-SP]

It is automatically determined when the menu key is depressed.

HSS AUTO	CARBIDE AUTO							

By selecting the material of the tool tip, the peripheral speed (/min) and feedrate (/rev) can be displayed (automatically determined).

⑪ Feed [FR]

Like peripheral speed, depression of the menu button automatically determines this item.

⑫ M code [M]

Data will enter when selection is made with a menu button or setting is made with a numerical key. (Two digits can be put in the former M and three digits can be entered in the latter M.)

01 OPT STOP	03 SPNDL FWD	04 SPNDL REV	05 SPNDL STOP	07 MIST COOLANT	08 FLOOD COOLANT	09 OFF COOLANT	50 AIR BLAST	NEXT (1/3)

00 PROG STOP	19 SPNDL ORIENT	35 T-BRK DETECT	38 SPNDL GEAR L/M	39 SPNDL GEAR H	33 OUT MSR-UNIT	34 IN MSR-UNIT		NEXT (2/3)

16 OPEN DRUM CVR	15 CLOSE DRUM CVR	11 TOOL UNCLAMP	10 TOOL CLAMP	53 CHIP VAC ON	09 CHIP VAC OFF			NEXT (3/3)

(3) Shape sequence data

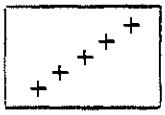
☐ Point machining shape

When the machining unit has been determined and the tool sequence data has been set, the machining shape must be set.
The point machining shape has the following seven patterns:

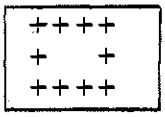
o Shape pattern list

I Point (PT)

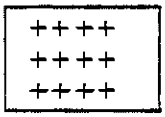
II Line (LIN)



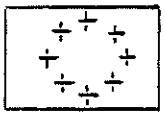
III Square (SQR)



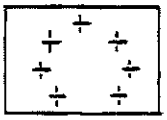
IV Grid (GRD)



V Circle (CIR)



VI Circular arc (ARC)



VII Chord (CHD)

Menu key	PT +

Fig. 1

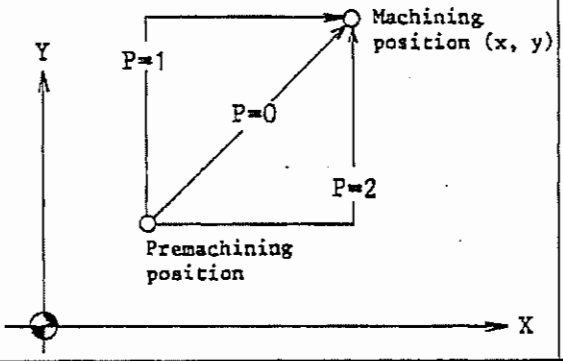
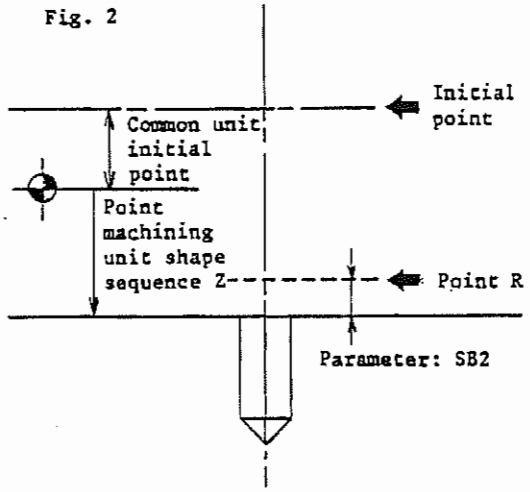


Fig. 2



Z	X	Y	AN1	AN2	T1	T2	F	M	N	P	Q	R
z	x	y	◆	◆	◆	◆	◆	◆	◆	0	0	0

Z : Z coordinate value on face to be machined (z)

X : X coordinate value of hole to be machined (x)

Y : Y coordinate value of hole to be machined (y)

P : Tool route pattern (Tool moves at 0 thru 2 as Fig. 1.)

Q : Set value should be 0 or 1.

1: Starting point (X,Y) is used only for positioning. At that point, no hole is bored.

0: Hole is also bored at the starting point (X, Y).

R : Return point level (See Fig. 2.)

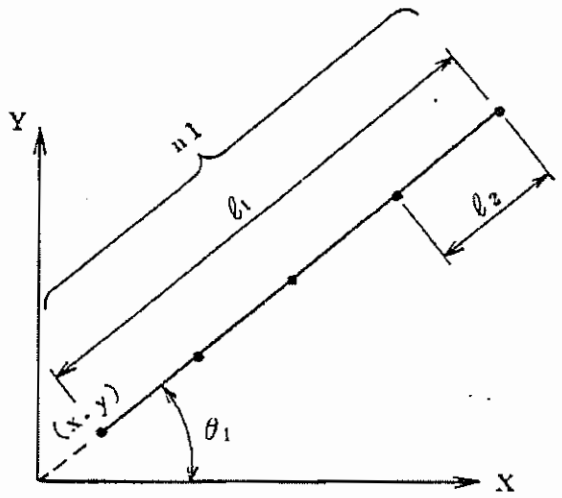
1: R point (point at a clearance of "parameter: BS2" from machined surface) return.

0: Initial point (defined in the workpiece coordinate system for common units) return.

For z subsequent shapes are identical in every pattern.

Note: When 1 or 2 is inputted in the first point P of the sequence data, processing is performed as if 0 were inputted.

Menu key	LIN
	+ + + +



Z	X	Y	AN1	AN2	T1	T2	F	M	N	P	Q	R
z	x	y	θ_1	◆	ℓ	◆	1	n1	◆	◆	0	0

Z : Z-coordinate value (Z) of machined surface.

X : X-coordinate value of first hole to be machined (x)

Y : Y-coordinate value of first hole to be machined (y)

AN1: Angle formed between x axis and straight line interconnecting holes to be machined (θ_1)

[CCW direction: +
CW direction : −]

T1: ℓ_1 = length between first and last holes to be machined (F = 1)

ℓ_2 = pitch between holes to be machined (F = 0)

F : Specifies ℓ, (To be set to 0 or 1.)

M : Number of holes to be machined (n1)

Q : To be set to 0 or 1.

With Q = 1, no drilling is performed at the starting point (x, y) where positioning only is done.

With Q = 0, drilling is performed also at the starting point (x, y).

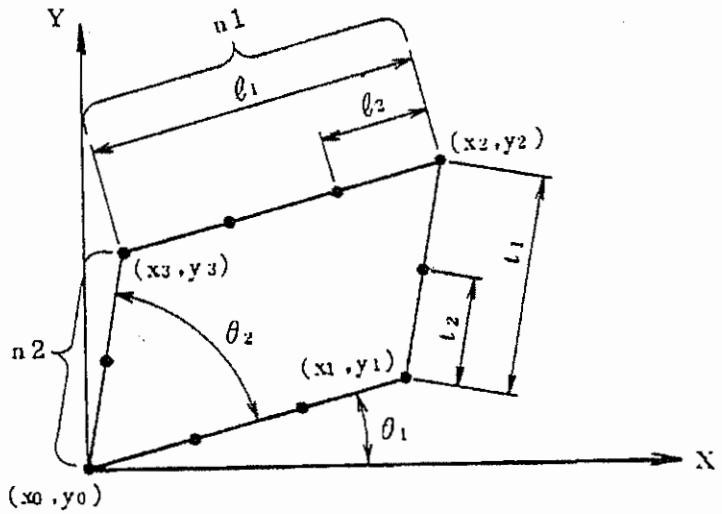
R : Return point level

1: R point (point at a clearance of "parameter: BS2" from machined surface) return.

0: Initial point (defined in the workpiece coordinate system for common units) return.

Menu key

SQR
++++
+ +
++++



Z	X	Y	AN1	AN2	T1	T2	F	M	N	P	Q	R
z	x	y	θ_1	θ_2	ℓ	t	1	n1	n2	0	0	0

Z : Z coordinate value (Z) of machined surface

X : X coordinate value of first hole to be machined (x)

Y : Y coordinate value of first hole to be machined (y)

AN1: Angle formed between x-axis and straight line of holes to be machined. (θ_1)

[CCW direction: +
CW direction: −]

AN2: Angle formed between straight lines of first and last holes to be machined. (θ_2)

[CCW direction: +
CW direction : −]

T1: Length of initially machined hole straight line ℓ1 (F = 1)
pitch of holes on initially machined hole straight line ℓ2 (F = 0)

T2: Length of last machined hole straight line t1 (F = 1)
pitch of holes on last machined hole straight line t2 (F = 0)

F : Specifies ℓ and t.

M : Number of holes on initially machined hole straight line: n1

N : Number of holes last machined hole straight line: n2

P : To be set to 0 or 1.

With P = 1, no hole is drilled at four corner points

(x_0, y_0), (x_1, y_1), (x_2, y_2) and (x_3, y_3)

With P = 0, holes are drilled also at four corner points.

(The drilling of starting point varies with setting 0.)

Q : To be set to 0 or 1

With Q = 1, positioning only is done at the starting point.

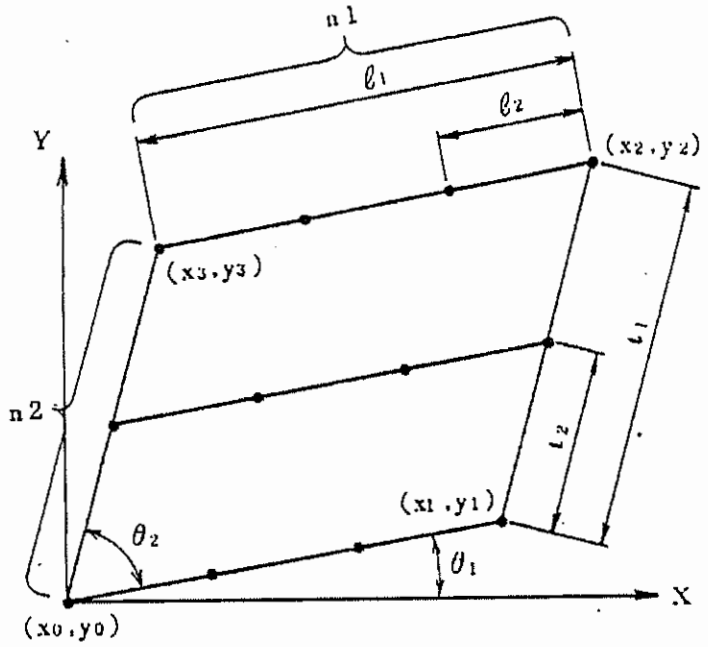
With Q = 0, the hole is drilled at the starting point.

R : Return point level

1: R point (point at a clearance of "parameter: BS2" from machined surface) return.

0: Initial point (defined in the workpiece coordinate system for common units) return.

Menu key	GRD ++++ ++++ ++++



Z	X	Y	AN1	AN2	T1	T2	F	M	N	P	Q	R
z	x	y	θ_1	θ_2	ℓ	t	1	n1	n2	0	0	0

Z : Z coordinate value (Z) of machined surface

X : x coordinate value of first hole to be machined (x)

Y : y coordinate value of first hole to be machined (y)

AN1: Angle formed between x-axis and straight line of holes to be machined (θ_1) [CCW direction: +, CW direction : -]

AN2: Angle formed between straight lines of first and last holes to be machined (θ_2) [CCW direction: +, CW direction : -]

T1: Length of initially machined hole straight line ℓ1 (F = 1)
pitch of holes on initially machined hole straight line ℓ2 (F=0)

T2: Length of last machined hole straight line t1 (F = 1)
pitch of holes on last machined hole straight line t2 (F=0)

F : Specifies ℓ and t.

M : Number of holes on initially machined hole straight line: n1

N : Number of holes on last machined hole straight line: n2

P : To be set to 0 or 1.

With P = 1, no hole is drilled at four corner points (x_0, y_0), (x_1, Y_1), (x_2, y_2) and (x_3, y_3).

With P = 0, holes are drilled also at four corner points. (The drilling of starting point varies with setting 0.)

Q : To be set to 0 or 1

With Q = 1, positioning only is done at the starting point.

With Q = 0, the hole is drilled at the starting point.

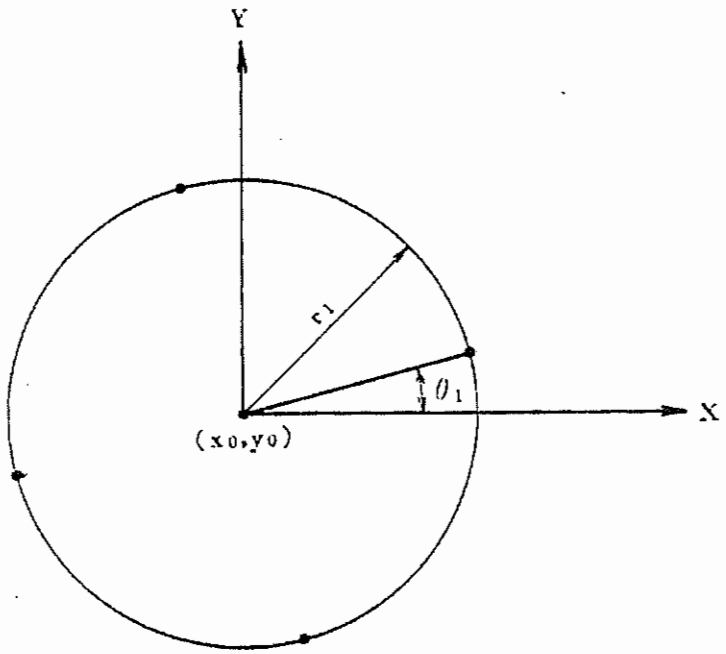
R : Return point level

1: R point (point at a clearance of "parameter: BS2" from machined surface) return.

0: Initial point (defined in the workpiece coordinate system for common units) return.

Menu key

CIR



Z	X	Y	AN1	AN2	T1	T2	F	M	N	P	Q	R
z	x	y	θ_1	◆	r1	◆	◆	n1	◆	◆		0

Z : Z coordinate value of machine surface (Z)

X : X coordinate value of circle center: x_0

Y : Y coorcinate value of circle center: y_0

AN1: Angle formed between initially machined hole and X-axis: θ_1 [CCW direction: +, CW direction : -]

T1 : Radius of circle : r1

M : Number of holes,machined: n1

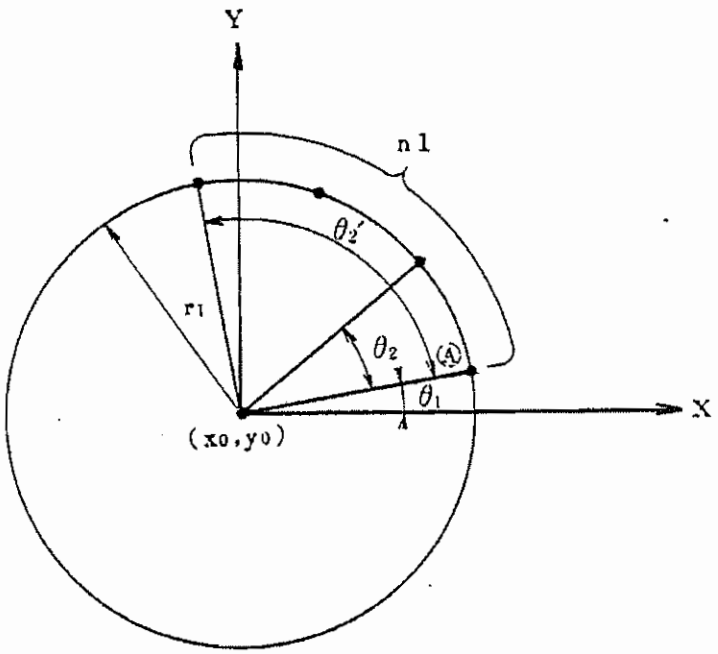
R : Return point level

1: R point (point at a clearance of "parameter: BS2" from machined surface) return.

0: Initial point (defined in the workpiece coordinate system for common units) return.

Menu key

ARC



Z	X	Y	AN1	AN2	T1	T2	F	M	N	P	Q	R
z	x	y	θ_1	θ_2	r1	◆	1	n1	◆	◆	1	0

Z : Z coordinate value of machined surface (Z)

X : X coordinate value of circle center: x_0

Y : Y coordinate value of circle center: y_0

AN1: Angle formed between initially machined hole (A) and X-axis: θ_1 [CCW direction: +, CW direction : −]

AN2: θ'_2 = Angle between initially and last machined holes (θ_2')
(F = 1) [CCW direction: +, CW direction : −]
θ_2 = Angle between adjacent machined holes (θ_2)
(F = 0) [CCW direction: +, CW direction : −]

T1 : Radius of circle: r1

F : Specifies θ_1 (to be set to 0 or 1).

M : Number of holes machined

Q : To be set to 0 or 1
With Q = 0, the hole is drilled at point (A).
With Q = 1, no hole is drilled at point (A).

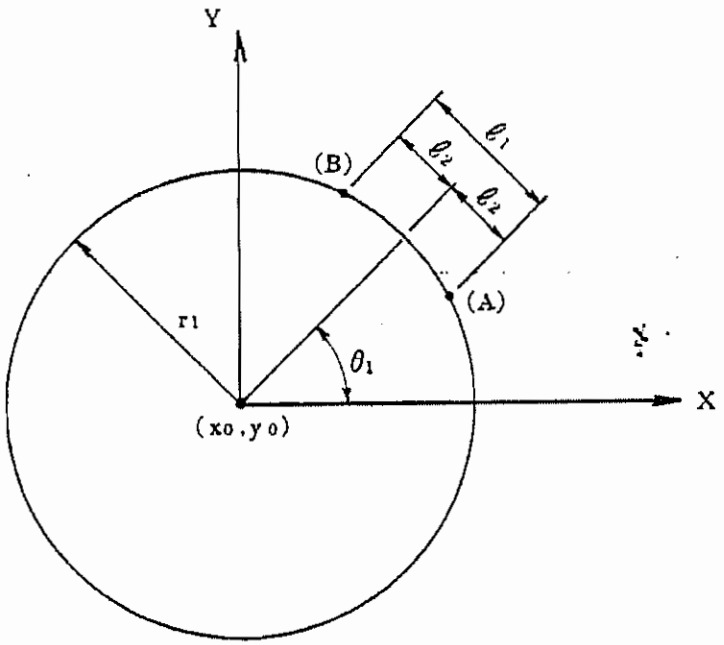
R : Return point level

1: R point (point at a clearance of "parameter: BS2" from machined surface) return.

0: Initial point (defined in the workpiece coordinate syatem for common units) return.

Menu key

CHD



Z	X	Y	AN1	AN2	T1	T2	F	M	N	P	Q	R
z	x	y	θ_1	◆	r_1	ℓ_1	◆	◆	◆	1		0

Z : Z coordinate value of machined surface (Z)

X : X coordinate value of circle center: x_0

Y : Y coordinate value of circle center: y_0

AN1: Angle formed between center line of chord and X-axis: θ_1 [CCW direction: +, CW direction : -]

T1 : Radius of circle: r_1

T2 : ℓ_1 = Pitch of machined holes or
ℓ_2 = Distance from center line, depending upon what P is taken

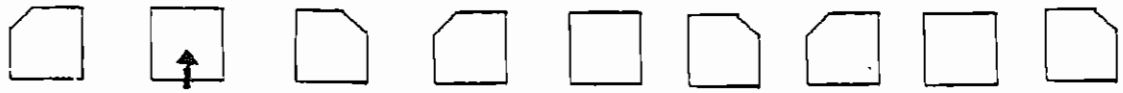
P : To be set to 0, 1 or 2
With P = 0, both (A) and (B) are machined. (Set ℓ_1 in T2)
With P = 1, (B) only is machined. (Set ℓ_2 in T2)
With P = 2, (A) only is machined. (Set ℓ_2 in T2)

R : Return point level
1: R point (point at a clearance of "parameter: BS2" from machined surface) return.
0: Initial point (defined in the workpiece coordinate system for common units) return.

2.1.4 Programming the line machining mode unit

Depress the LINE/MACH-ING mode menu.

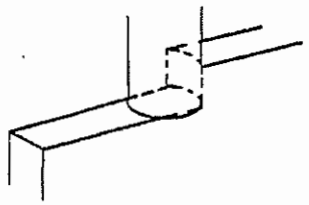
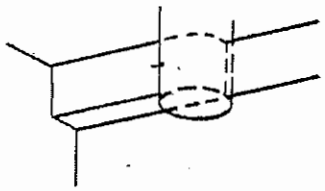
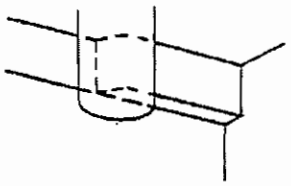
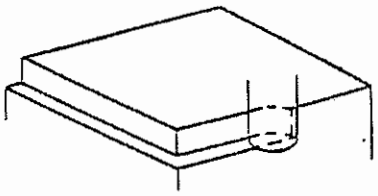
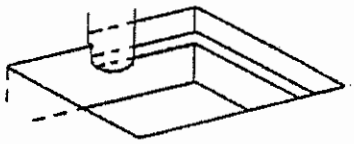
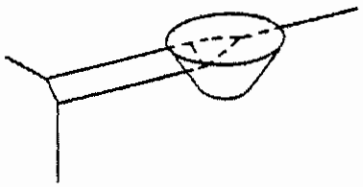
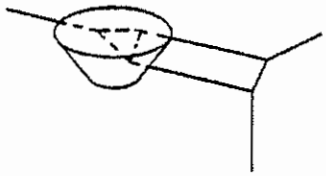
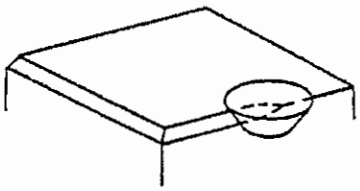
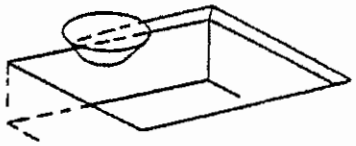
POINT MACH-ING	LINE MACH-ING	FACE MACH-ING	MANUAL PROGRAM	OTHER	WPC	OFFSET	END	GROUP CHECK



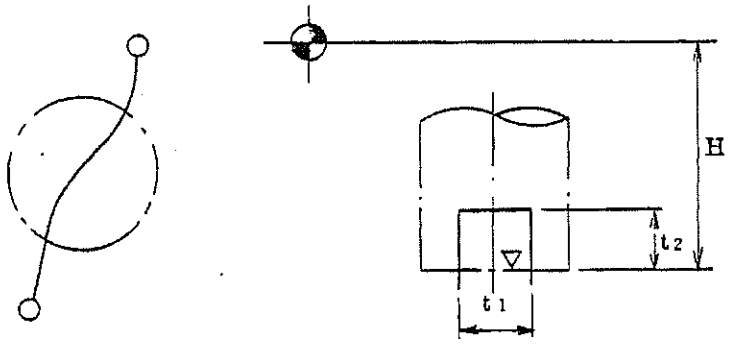
Once the menu has been changed over to the machining unit selection menu, select the related unit.

(1) Unit data

LINE CTR	LINE RGT	LINE LFT	LINE OUT	LINE IN	CHMF RGT	CHMF LFT	CHMF OUT	CHMF IN

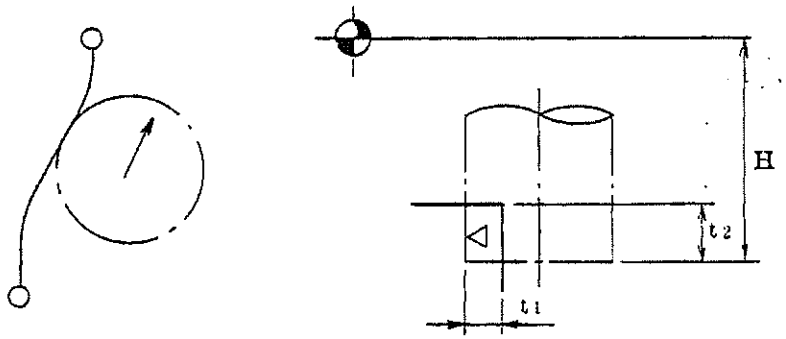
a. Line machining center 	b. Line machining right 	e. Line machining left 
d. Line machining out 	e. Line machining in 	g. Chamfering right 
h. Chamfering left 	i. Chamfering out 	j. Chamfering in 

Unit Name	LINE CTR
Menu Key	LINE CTR



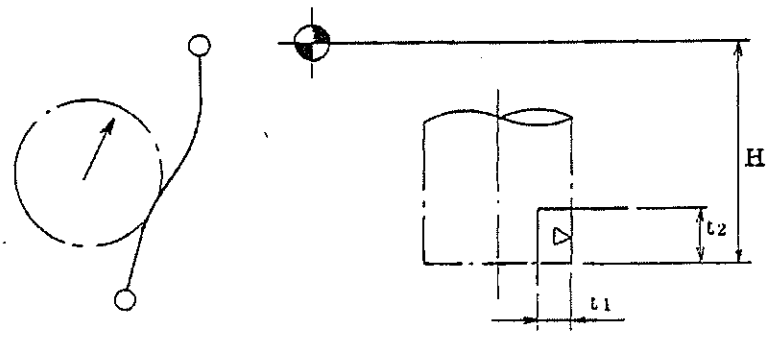
UNIT	DEPTH(H)	SRV-Z(t_2)	SRV-R(t_1)	RGH(▽)	CHMF	FIN-Z	FIN-R
LINE CTR (M)	999.999	99.999	99.999	9 (M)	◆	99.999 (AUTO)	◆

Unit Name	LINE RGT
Menu Key	LINE RGT



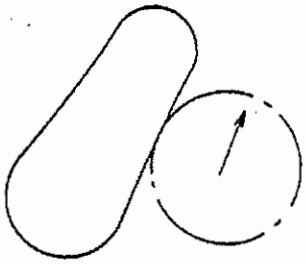
UNIT	DEPTH(H)	SRV-Z(t_2)	SRV-R(t_1)	RGH(▽)	CHMF	FIN-Z	FIN-R
LINE RGH (M)	999.999	99.999	99.999	9 (M)	◆	99.999 (AUTO)	99.999 (AUTO)

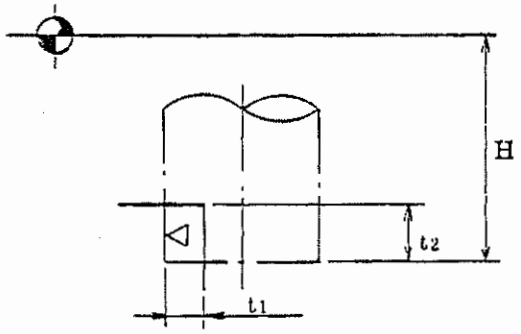
Unit Name	LINE LFT
Menu Key	LINE LFT



UNIT	DEPTH(H)	SRV-Z(t_2)	SRV-R(t_1)	RGH(▽)	CHMF	FIN-Z	FIN-R
LINE LFT (M)	999.999	99.999	99.999	9 (M)	◆	99.999 (AUTO)	99.999 (AUTO)

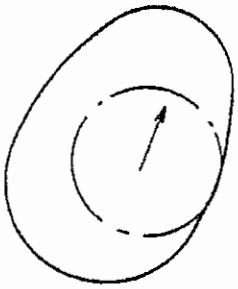
Unit Name	LIN OUT
Menu Key	LINE OUT

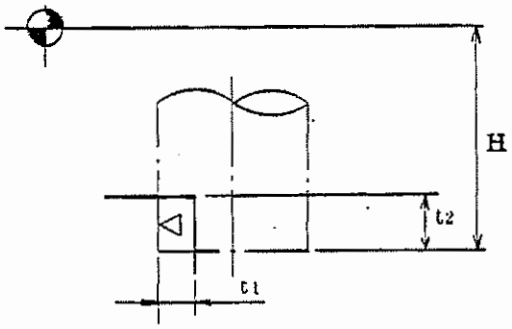




UNIT	DEPTH(H)	SRV-Z(t_2)	SRV-R(t_1)	RGH(▽)	CHMF	FIN-Z	FIN-R
LINE OUT (M)	999.999	99.999	99.999	9 (M)	◆	99.999 (AUTO)	99.999 (AUTO)

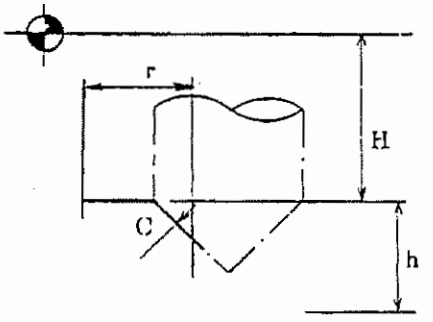
Unit Name	LIN IN
Menu Key	LINE IN





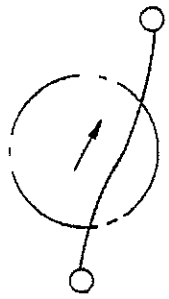
UNIT	DEPTH(H)	SRV-Z(t_2)	SRV-R(t_1)	RGH(▽)	CHMF	FIN-Z	FIN-R
LINE IN (M)	999.999	99.999	99.999	9 (M)	◆	99.999 (AUTO)	99.999 (AUTO)

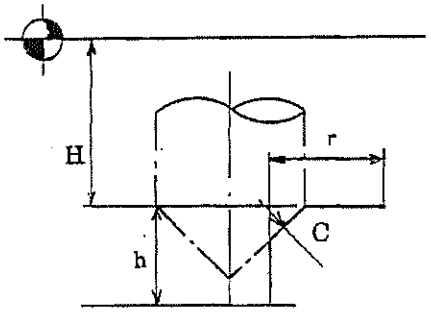
Unit Name	CHMF RGT
Menu Key	CHMF RGT



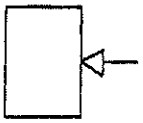
UNIT	DEPTH(H)	INTER-Z(h)	INTER-R(r)	RGH(▽)	CHMF(C)
CHMF RGT (M)	999.999	99.999	99.999	◆	99.9

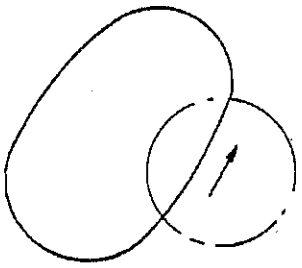
Unit Name	CHMF LFT
Menu Key	CHMF LFT 

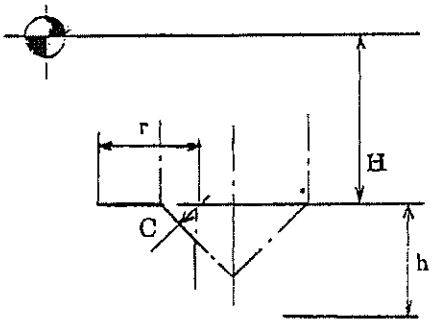




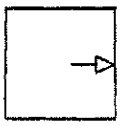
UNIT	DEPTH(H)	INTER-Z(h)	INTER-R(r)	RGH(▽)	CHMF(C)
CHMF LFT (M)	999.999	99.999	99.999	◆	99.9

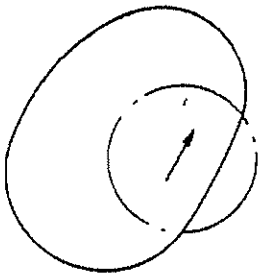
Unit Name	CHMF OUT
Menu Key	CHMF OUT 

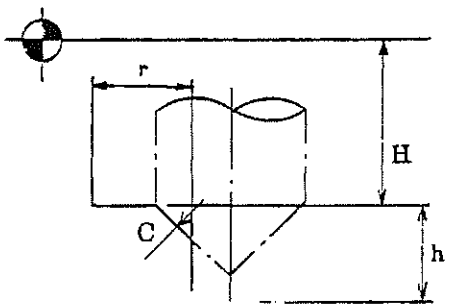




UNIT	DEPTH(H)	INTER-Z(h)	INTER-R(r)	RGH(▽)	CHMF(C)
CHMF OUT (M)	999.999	99.999	99.999	◆	99.9

Unit Name	CHMF IN
Menu Key	CHMF IN 





UNIT	DEPTH(H)	INTER-Z(h)	INTER-R(r)	RGH(▽)	CHMF(C)
CHMF IN (M)	999.999	99.999	99.999	◆	99.9

(2) Tool Sequence Data

End mill, face mill, chamfering cutter and ball end mill

SNO	TOOL	NOM-ϕ	NO	APRCH-X	APRCH-Y	TYPE	ZFD	DEP-Z	WID-R	C-SP	FR	M	N
R1 (F1)	①	②	③	④	⑤	⑥	⑦	⑧	⑨	⑩	⑪	⑫	⑫

① Tool name (automatically determinad) [TOOL]

Once a tool development has been done, E-MILL(endmill) is automatically displayed.

ENDMILL	FACE MILL	CHAMF CUTTER	BALL ENDMILL				

② Nominal tool diameter [NOM-ϕ]

Use ten keys to input the nominal diameter of a tool.

Use a menu key to identify identical tool

A	B	C	D	E	F	G	H	J

Unless the nominal tool diameter has been registered in the tool file beforehand, an alarm will result.

③ Tool priority machining Nos. [NO.]

Proceed to input of tool priority machining numbers (prior machining No./subsequent machining No.).
For input of machining numbers, menu keys and/or ten keys are used:

(1) Prior machining No.
It is input with ten keys only. (1-63)

(2) Subsequent machining No.
The DELAY PRIORITY menu key is depressed to inverse the menu. Then, the subsequent machining No. is entered with ten keys. (1-63)

(3) None (Input is unnecessary.)

Menu

	DELAY PRIORITY		PRI. NO. CHANGE	PRI. NO ASSIGN		PRI. NO ALL ERAS	SUB PRO= PROC END	

For details, see "2.2 Same Tool Priority Machining Function".

④ Tool approaching point coordinate X [APPR-X]

⑤ Tool approaching point coordinate Y [APPR-Y]

Use ten keys to input thise coordinates or depress menu key AUTO SET. (③ and ④ in common)

⑥ How to move a tool [TYPE]

Select CW or CCW cutting, using menu keys.
This function, however, is available only in the four units LINE OUT, LINE IN, CHMF OUT and CHMF IN.

⑦ Selection of feed in Z-axis direction [ZFD]

The feed speed at which the tool cuts from the approach point in the Z-axis direction is set in terms of a multiple of feed speed in the radius direction, using ten keys.

Otherwise, either G01 feed (30% of the radius direction feed rate)(see Note) or G00 feed (rapid feed) is selected with menu keys.

JOG G01	RAPID G00							

Note: Thirty percent is the standard value set by means of line and face machining parameters.

⑧ Z-axial depth of cut per cycle [DEP-Z]

With the AUTO SET key depressed, the Z-axial depth is automatically computed.
Ten keys can be also used to input the depth.
(To be inputted in the rough machining sequence only out of LINE CTR, LINE RGT, LINE LFT, LINE OUT and LINE IN.)

⑨ Diametric depth of cut per cycle [WID-R]

In line machining, this depth is not required to be inputted.

⑩ Peripheral speed [C-SP] ⑪ Feed [FR]

HSS	CARBIDE
AUTO	AUTO

Selecting a material of the tool cutting edge with a menu key will cause peripheral speed (/min) and feedrate (/rev) to be automatically determined.

Note : For ball end mill, it is not automatically determined.

⑫ M code [M]

Data can be inputted either by selecting with a menu key or by using ten keys. (The former M can be inputted in two digits while the latter can be inputted in three digits.)

(3) Shape Sequence Data

■ Line/Face machining

After determining a machining unit and inputting the tool sequence data, then input a machining shape

Line/Face machining shapes are available in the following patterns :

I SQUARE

II CIRCLE

III ARBITRY ... LINE CW CCW SHAPE SHIFT
SHAPE ROTATE(CW), SHAPE ROTATE(CCW)

Note : The arbitrary shape cannot cover every shape due to a CPU operation error processing. Do not fail to check with GRAPHIC, etc, before applying the ARBITRY pattern, accordingly.

Pattern selecting menu (I)

SQUARE	CIRCLE		ARBITRY				SHAPE END	CHECK

Pattern selecting menu (II) - In case of ARBITRY only

LINE	CW ARC 	CCW ARC 	SHAPE ROTATE	SHAPE SHIFT	REPEAT END	STATING POINT	SHAPE END	CHECK

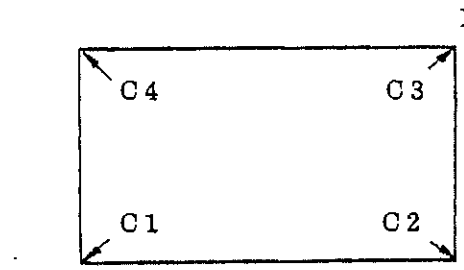
Pattern selectin menu (III) - In case of SHAPE ROTATION only

CW SHIFT 	CCW SHIFT 							

(i) FIX SHAPE SQUARE

Shape pattern select key

SQUARE



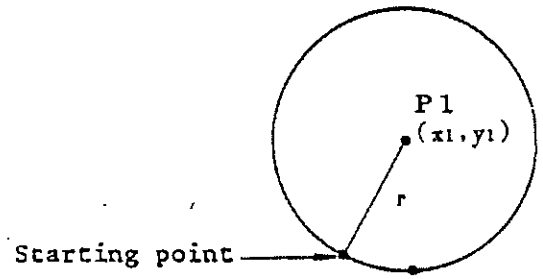
P1X/CX	P1Y/CY	P3X/R	P3Y	CN1	CN2	CN3	CN4
x_1	y_1	x_3	y_3	C_1 (R_1)	C_2 (R_2)	C_3 (R_3)	C_4 (R_4)

P1X/CX : x coordinate value of starting point : x1
P1Y/CY : y coordinate value of starting point : y1
P3X/R : x coordinate value of diagonal line : x1
P3Y : y coordinate value of diagonal line : y3
CNn : C chamfering at P_n or arc (R)

(ii) FIX SHAPE CIRCLE

Shape pattern select key

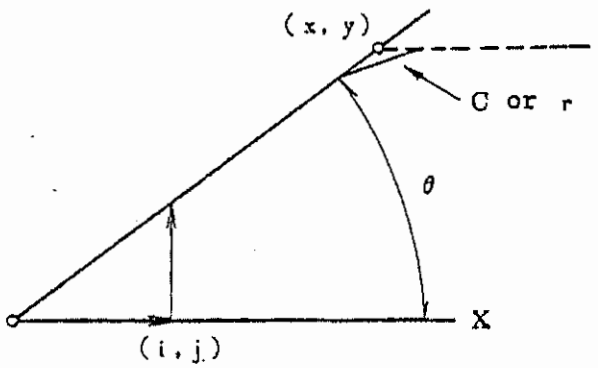
CIRCLE



P_1X/CX	P_1Y/CY	P_3X/R	P_3Y	CN_1	CN_2	CN_3	CN_4
x_1	y_1		◆	◆	◆	◆	◆

P1X/CX : x coordinate value of circle center : x_1
P1Y/CY : y coordinate value of circle center : y_1
P3X/R : radius of circle : r

(iii) ARBITRARY SHAPE

Shape pattern select key		
I	ARBITRY 	
II	LINE	

PTN	X	Y	R/θ	I	J	P	CNR
LINE	x	y	θ	i	j	p	C (r)

X : x coordinate value of end point: x

Y : y coordinate value of end point: y

R/θ : angle formed between straight line and x axis : θ [CCW direction +]

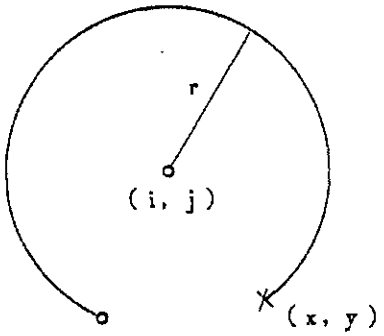
I : x-axial vector : i

J : y-axial vector : j

P : requirement for cross with next shape : p

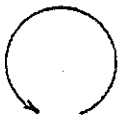
CNR : Seam chamfering or arc. : c(r)

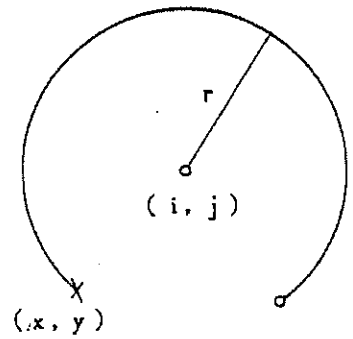
Shape pattern select key	
I	ARBITRY 
II	CW ARC 



PTN	X	Y	R/θ	I	J	P	CNR
CW	x	y	r	i	j	p	c

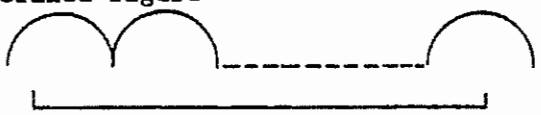
X : X coordinate value of end point : x
Y : Y coordinate value of end point : y
R/θ : radius of circle : r
I : X coordinate value of circle center : i
J : Y coordinate value of circle center : j
P : requirement for cross with next shape : p
CNR : Seam chamfering or arc. : c (r)

Shape pattern select key	
I	ARBITRY 
II	CCW ARC 



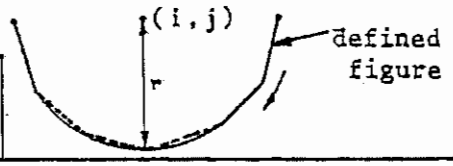
PTN	X	Y	R/θ	I	J	P	CNR
CCW	x	y	r	i	j	p	c

X : X coordinate value of end point : x
Y : Y coordinate value of end point : y
R/θ : radius of circle : r
I : X coordinate value of circle center: i
J : Y coordinate value of circle center: j
P : requirement for cross with next shape : p
CNR : Seam chamfering or arc : c(r)

Shape pattern select key			
	I	ARBITRY	defined figure  P pieces
	II	SHAPE SHIFT	

PTN	X	Y	R/θ	I	J	P	CNR
SHAPE SHIFT	◆	◆	◆	◆	◆	P	◆

P : Cycle of repeating the defined figure from SHAPE SHIFT to REPEAT END

Shape pattern select key				
	I	ARBITRY		
	II	SHAPE ROTATE	CW-SHIFT	

PTN	X	Y	R/θ	I	J	P	CNR
CW-SHIFT	◆	◆	r	i	j	p	◆

R/θ ; Radius r to rotate the defined figure
I : Center x coordinate i to rotate the defined figure
J : Center y coordinate j to rotate the defined figure
P : Cycles of repeating the defined figure

Shape pattern select key				
	I	ARBITRY		
	II	SHAPE ROTATE	CCW-SHIFT	

PTN	X	Y	R/θ	I	J	P	CNR
CCW SHIFT	◆	◆	r	i	j	p	◆

R/θ : Radius r to rotate the defined figure
I : Center X coordinate i to rotate the defined figure
J : Center Y coordinate j to rotate the defined figure
P : Cycles of repeating the defined figure

Arbitry pattern definition

Arbitry shapes are applicable to line and face machining.

The arbitry shape pattern is defined as follows :

(Note) Be desired shape machining function cannot serve for all the purposes because of the imitations of the error processing in operation of the CPU. Before utilizing the function, never fail to perform checking with GRAPHICS or similar means.

1) Program

SNO	PTN	X	Y	R/θ	I	J	P	CNR

Definition of titles used on the CRT

SNO : sequence number(to be serial in the unit)
PTN : ... LINE
... CW ARC
... CCW ARC

X,Y : coordinates of end point (However, input the starting point initially.)

R/θ : machining pattern element
angle formed with X axis for LINE
radius formed with X axis for ARC

I, J: machining pattern elements
line vector for LINE
arc center (i,j) for ARC

P : modifier (positioning relation between following figure and crossing point)
UP
DOWN
LEFT
RGT (right)
CANCEL (To cancel the modifier mentioned above)

CNR : specifies a corner R or C at end point

C ..... LINE-LINE only

R ..... between arbitry figures (which may not have any crossing point)

2) Precuations upon Entry

(1) How to use "?"

a. The fact that "?" has been entered and set is not identical with a skip due to a motion of the cursor in the sense that data are invalid.

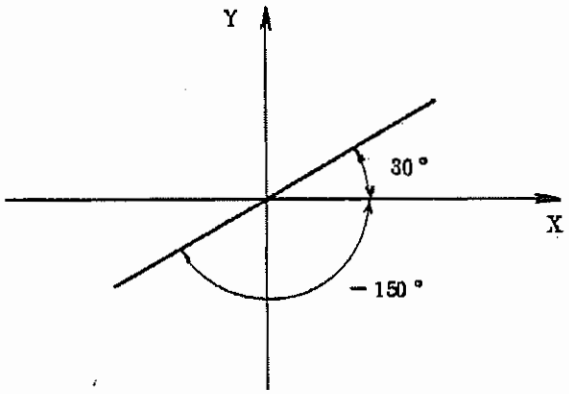
b. For the automatic programming results, only those items in which "?" is entered will be displayed. For end point X or Y, however, it is necessary to enter data or "?".

c. Line I or J will not be displayed even if "?" is entered. (No special internal calculation will be done.)

d. When a very long machining program is being executed which could cause a bubble transmission, the results of the measurement, is some cause, may not be displayed when "?" is inputted.

(2) All coordinates of an end point and of a circle enter are entered in an absolute coordinate system.

(3) The straight line defined with θ or I, J is infinite in both directions. (It is not semi-linear.) In other words, $\theta = 30°$ and $\theta = -150°$ represent an identical straight line as illustrated below.



(4) Words to be selected

If it is necessary to select a word for a crossing or contacting point, its entry is urgened in a message.(These words may be entered in advance.)

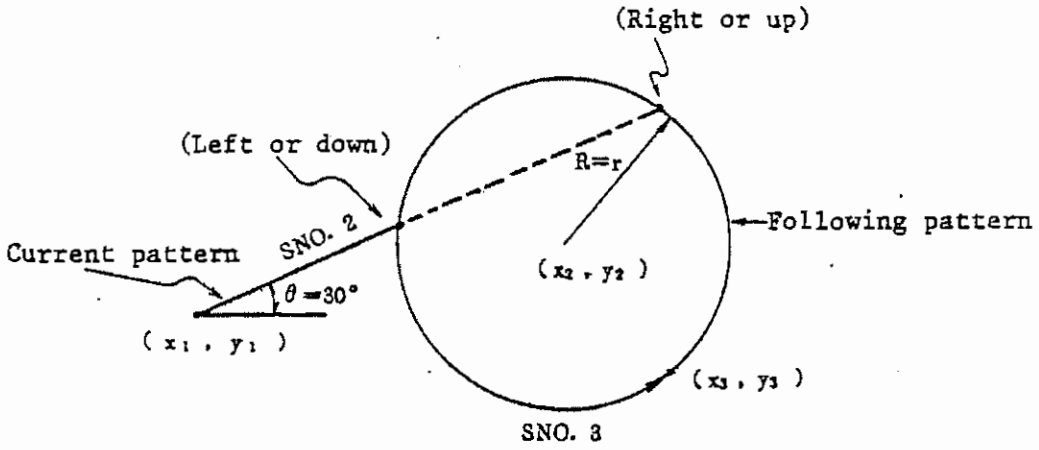
Example

UP	DOWN	LEFT	RIGHT

With cursor → shape pattern definition address P,

UP	DOWN	LEFT	RIGHT		CANCEL			

This menu is used to specify the point at which a shape pattern defined in the current sequence comes in contact with or crosses a shape pattern defined in the following sequence.



(Definition of a starting point)	FIG	ARBITRY PTN	X	Y	R/θ	I	J	P	CNR
	1	LINE	x_1	y_1					
Current	2	LINE	?	?	30			LEFT	
Following	3	CCW	x_3	y_3	r	x_2	y_2		

3) Figure Definition Pattern

No. Note 1)	PTN	X	Y	R/Θ	I	J	P	CNR/CNC
P - 1	LINE	○	○					
P - 2	LINE	○	?	○				
P - 3	LINE	?	○	○				
P - 4	LINE	○	?	?/→	○	○		
P - 5	LINE	?	○	?/→	○	○		
P - 6	CW/CCW	○	○	○				
P - 7	CW/CCW	○	○	?/→	○	○		
P - 8	CW/CCW	?	○	?/→	○	○	○	
P - 9	CW/CCW	○	?	?/→	○	○	○	
P - 10	CW/CCW	○	○	?/→	?	○		
P - 11	CW/CCW	○	○	?/→	○	?		
Q - 1	LINE	?	?	○			(○)	△
Q - 2	LINE	?	?	?/→	○	○	(○)	△
Q - 3	LINE	?	?	?/→	?/→	?/→	(○)	△
Q - 4	CW/CCW	?	?	?/→	○	○	(○)	△
Q - 5	CW/CCW	?	?	○	?/→	?/→	○	△
R - 1	LINE	○	○	○			(○)	△
R - 2	CW/CCW	○	○	○	○	○	(○)	△
R - 3	CW/CCW	○	?	○	○	○	(○)	△
R - 4	CW/CCW	?	○	○	○	○	(○)	△
R - 5	CW/CCW	?	?	○	○	○	(○)	△
R - 6	LINE	○	○	?/→	○	○	(○)	△

Note 1. For each of the figure definition patterns, take note of following:

P1 - P11: The figure definition pattern can be terminated in the form of this input data.

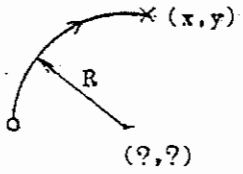
Q1 - Q5 : Data are insufficient and a support with next figures is required.

R1 - R6 : The excess data which have been entered will be required to define figures before and after.

In the table above, "○" represents an item to be entered.
"?" represents a "?" input.
"?/→" represents a "?" input or a skip due to a cursor motion "→".
"(○)" may need input as the case may be.
"△" Input to specify the corner.

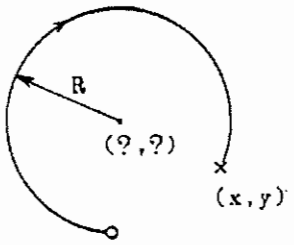
P - 6

With R > 0

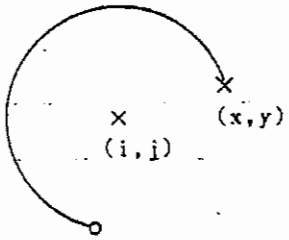


P - 6

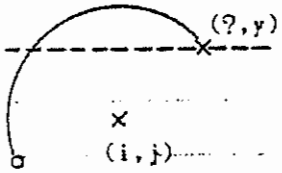
With R < 0



P - 7

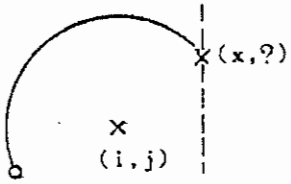


P - 8



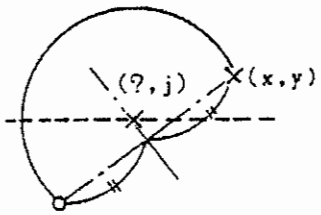
Modifier P is used to specify RICHT (or LEFT).

P - 9

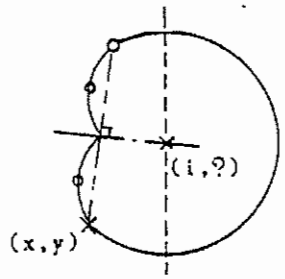


Modifier P is used to specify UP (or DOWN).

P-10

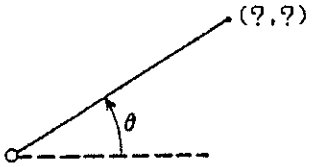


P - 11



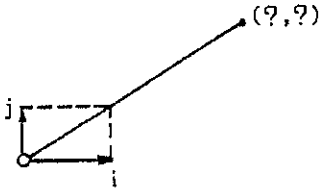
Q - 1

With P specified



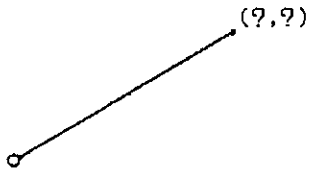
Q - 2

With P specified



Q - 3

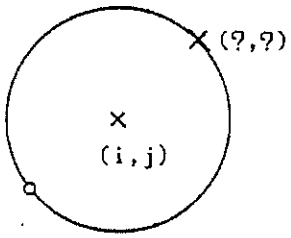
With P specified



However, definition will be possible only when the line comes in contact with a next arc.
(I, J and R are required.)

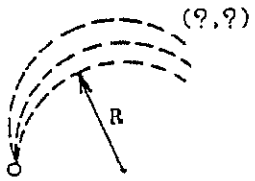
Q - 4

With P specified



Q - 5

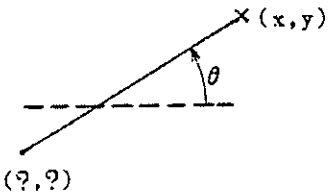
With P specified



However, definition will be possible only when the line comes in contact with a next arc.
(LINE(X,Y, Θ) or ARC (I, J, R) is required.

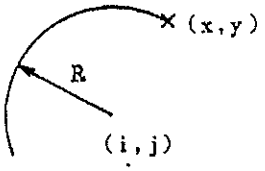
R - 1

With P specified



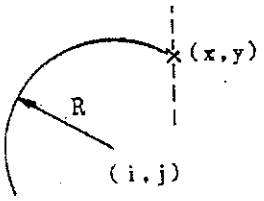
R - 2

With P specified



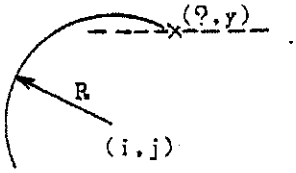
R - 3

With P specified



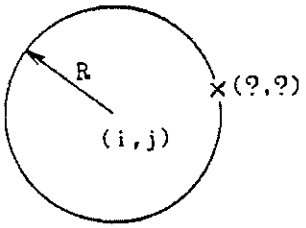
R - 4

With P specefied



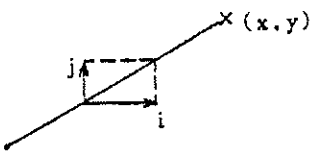
R - 5

With P specified



R - 6

With P specified



4) Pattern Combination

Forerunning Pattern \ Following Pattern	P1	P2	P3	P4	P5	P6	P7	P8	P9	P10	P11	Q1	Q2	Q3	Q4	Q5	R1	R2	R3	R4	R5	R6
P 1	○	○	○	○	○	○	○	○	○	○	○	○	○	○	○	○	△	△	△	△	△	△
P 2	○	○	○	○	○	○	○	○	○	○	○	○	○	○	○	○	△	△	△	△	△	△
P 3	○	○	○	○	○	○	○	○	○	○	○	○	○	○	○	○	△	△	△	△	△	△
P 4	○	○	○	○	○	○	○	○	○	○	○	○	○	○	○	○	△	△	△	△	△	△
P 5	○	○	○	○	○	○	○	○	○	○	○	○	○	○	○	○	△	△	△	△	△	△
P 6	○	○	○	○	○	○	○	○	○	○	○	○	○	○	○	○	△	△	△	△	△	△
P 7	○	○	○	○	○	○	○	○	○	○	○	○	○	○	○	○	△	△	△	△	△	△
P 8	○	○	○	○	○	○	○	○	○	○	○	○	○	○	○	○	△	△	△	△	△	△
P 9	○	○	○	○	○	○	○	○	○	○	○	○	○	○	○	○	△	△	△	△	△	△
P 10	○	○	○	○	○	○	○	○	○	○	○	○	○	○	○	○	△	△	△	△	△	△
P 11	○	○	○	○	○	○	○	○	○	○	○	○	○	○	○	○	△	△	△	△	△	△
Q 1	×	×	×	×	×	○	○	×	×	×	×	×	×	×	×	×	○	○	○	○	○	○
Q 2	×	×	×	×	×	○	○	×	×	×	×	×	×	×	×	×	○	○	○	○	○	○
Q 3	×	×	×	×	×	×	○	×	×	×	×	×	×	×	×	×	×	○	○	○	○	×
Q 4	○	×	×	×	×	○	○	×	×	×	×	×	×	×	○	×	○	○	○	○	○	○
Q 5	×	×	×	×	×	×	×	×	×	×	×	×	×	×	×	×	×	○	○	○	○	(○)
R 1	○	○	○	○	○	○	○	○	○	○	○	○	○	○	○	○	△	△	△	△	△	△
R 2	○	○	○	○	○	○	○	○	○	○	○	○	○	○	○	○	△	△	△	△	△	△
R 3	○	○	○	○	○	○	○	○	○	○	○	○	○	○	○	○	△	△	△	△	△	△
R 4	(○)	×	(○)	×	(○)	○	○	○	○	○	○	×	×	×	○	○	×	△	△	△	△	×
R 5	○	×	×	×	×	×	○	×	×	×	×	×	×	×	×	×	○	○	○	○	○	○
R 6	○	○	○	○	○	○	○	○	○	○	○	○	○	○	○	○	△	△	△	△	△	△

Symbols used in the table above have the following meanings:

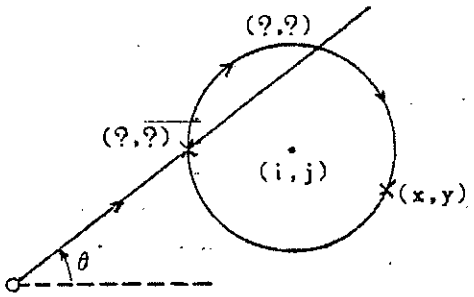
- o : Definable
- Δ : Definable (but with too much information)
- × : Undefinable

5) Q Group Forerunning Pattern Followed by a Definable Pattern

(1) Forerunning pattern: Q-1

a. Q-1, P-6

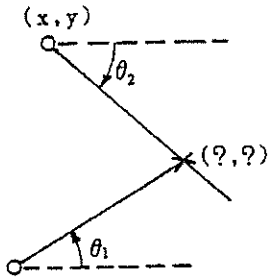
a. Q-1, P-6



Example of Input

PTN	X	Y	R/θ	I	J	P
LINE	?	?	θ			0
CW	X	y		i	j	

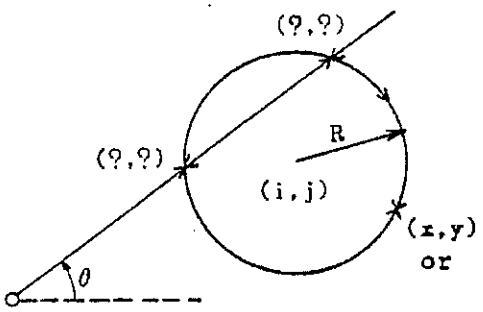
b. Q-1, R-1



Example of Input

	X	Y	R/θ	I	J	P
Q-1	?	?	θ_1			
R-1	x	y	θ_2			

c. Q-1, R-2 (or R-3, R-4, R-5)



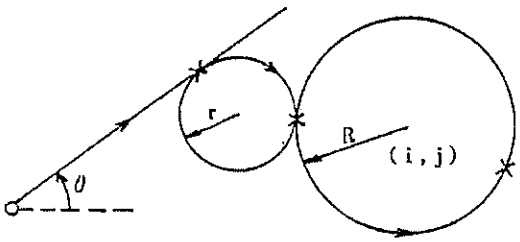
or (x,?),(?,y)

Example of Input

PTN	X	Y	R/θ	I	J	P
LINE	?	?	θ			0
CW	x/?	y/?	R	i	j	

With modifier P specified, right or left crossing point is selected.

If no crossing point exists, denifition is impossible. However, it will be possible, with an appropriate corner R inserted.

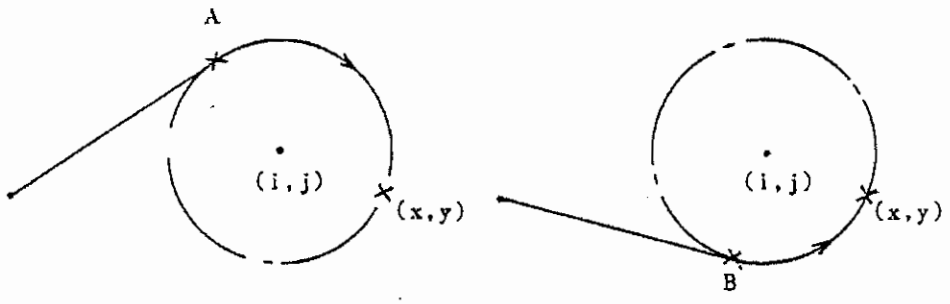


(2) Forerunning pattern: Q-2

Similar to the case of Q-1, with Arctan (J/I) = θ

(3) Forerunning pattern: Q-3

a. Q-3, P-7 (or R-2, R-3, R-4, R-5)



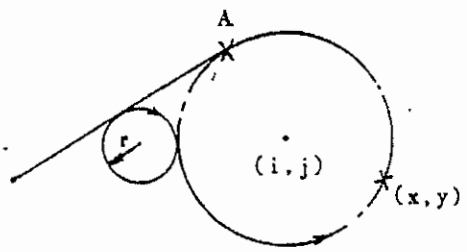
Without modifier P specified, a path will be automatically selected according to the turning direction of the arc.

A ..... Arc is turning CW.

B ..... Arc is turning CCW.

In case where CCW is selected as the turning direction of the arc and contacting point A is required, UP (or LEFT) should be selected as modifier P.

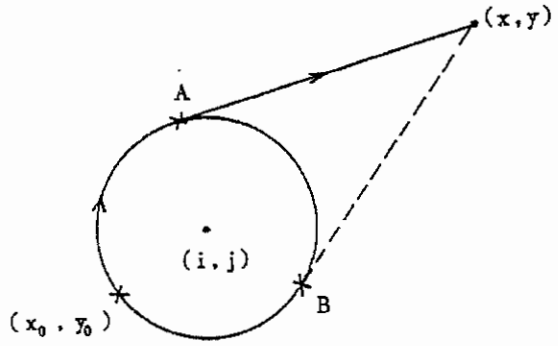
In this case, corner R can be specified.



CCW

(4) Forerunning pattern: Q-4

a. Q-4, P-1



Example of Input

PTN	X	Y	R/θ	I	J	P
CW	?	?	?	i	j	→
LINE	x	y				

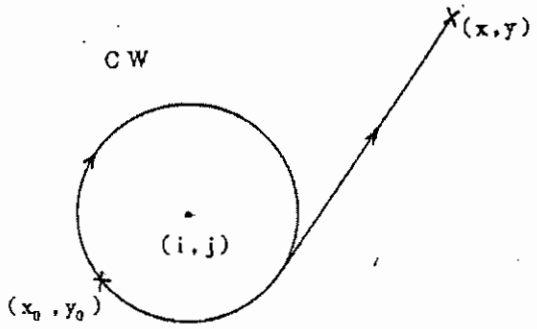
Without modifier P specified, a path will be automatically selected according to the turning direction of the arc.

A ..... Arc is turning CW.

B ..... Arc is turning CCW.

An instance with modifier P specified is shown below.

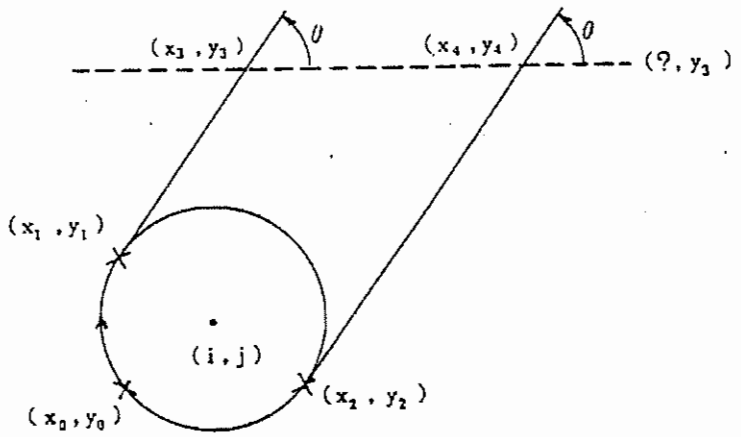
CW modifier P = DOWN



Example of Input

PTN	X	Y	R/θ	I	J	P
CW	?	?	?	i	j	DOWN
LINE	x	y				

b. Q-4, P-3 (or P-5)



A table given below shows the relation between an example of input and a selected path

	Example of Input							Path Selected							
	P T N	X	Y	R/θ	I	J	P	P T N	X	Y	R/θ	I	J	P	Path
Ex. 1	CW	?	?	?	i	j	→	CW	x_1	y_1		i	j	→	
	LINE	?	x_3	θ				LINE	x_3	y_3	θ				
Ex. 2	CCW	?	?	?	i	j	→	CCW	x_2	y_2		i	j	→	
	LINE	?	y_3	θ				LINE	x_4	y_3	θ				
Ex. 3	CW	?	?	?	i	j	DOWN	CW	x_2	y_2		i	j	DOWN	
	LINE	?	y_3	θ				LINE	x_4	y_3	θ				
Ex. 4	CCW	?	?	?	i	j		CCW	x_1	y_1		i	j	UP	
	LINE	?	y_3	θ			UP	LINE	x_3	y_3	θ				

In the table above, modifier P has the following meanings:

P = ⌜→⌟ : skipped with the cursor

P = UP : UP specified

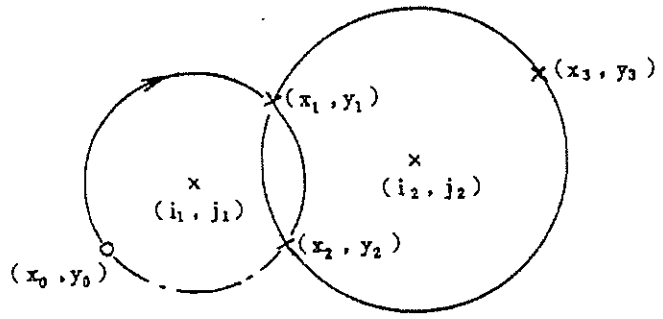
P = DOWN : DOWN specified

c. Q-4, P-2 (or P-4)

Similar to the case of preceding b. Q-4, P-3 (or P-5) but (?, y3) replaced with (x3,?).

d. Q-4, P-7

In this combination, both arcs are determined. With modifier P specified, therefore, a crossing point is selected. Unless any crossing point exists, definition will be possible, with both arcs connected by inserting an appropriate corner R.



A table below shows the relation between examples of inputs and paths selected.

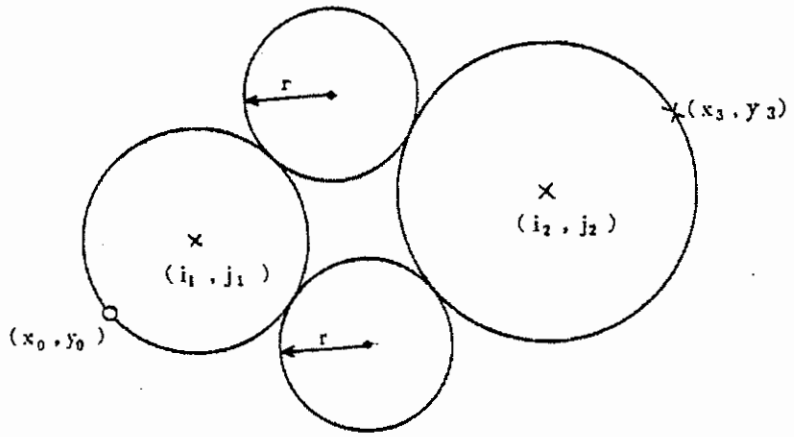
	Example of Input							Path selected						
	PTN	X	Y	R/θ	I	J	P	PTN	X	Y	R/θ	I	J	P	Path
Ex.1	CW	?	?	?	i_1	j_1	UP	CW	X_1	Y_1		i_1	j_1	UP	
	CCW	X_3	Y_3		i_2	j_2		CCW	X_3	Y_3		i_2	j_2		
Ex2.	CW	?	?	?	i_1	j_1	UP	CW	X_1	Y_1		i_1	j_1	UP	
	CW	X_3	Y_3		i_2	j_2		CW	X_3	Y_3		i_2	j_2		
Ex.3	CW	?	?	?	i_1	j_1	DOWN	CW	X_2	Y_2		i_1	j_1	DOWN	
	CCW	X_3	Y_3		i_2	j_2		CCW	X_3	Y_3		i_2	j_2		
Ex.4	CW	?	?	?	i_1	j_1	DOWN	CW	X_2	Y_2		i_1	j_1	DOWN	
	CW	X_3	Y_3		i_2	j_2		CW	X_3	Y_3		i_2	j_2		

In the table above, modifier P has the following meaning:

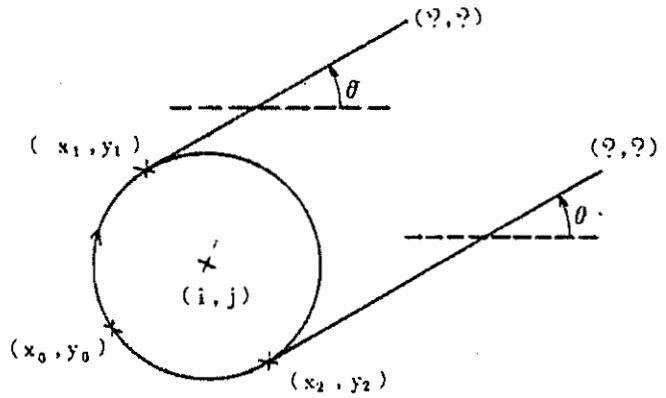
P = UP: UP specified

P = DOWN: DOWN specified

r>0 Corner R contacts in CW direction
r<0 Corner R contacts in CCW direction



e. Q − 4 , Q − 1 (or Q-2)



A table below shows the relation between example of input and path.

	Example of Input							Path selected							
	PTN	X	Y	R/θ	I	J	P	PTN	X	Y	R/θ	I	J	P	Path
Ex. 1	CW	?	?	?	i	j	→	CW	x_1	y_1		i	j	→	
	LINE	?	?	θ				LINE	?	?	θ				
Ex. 2	CCW	?	?	?	i	j	→	CCW	x_2	y_2		i	j	→	
	LINE	?	?	θ				LINE	?	?	θ				
Ex. 3	CW	?	?	?	i	j	DOWN	CW	x_2	y_2		i	j	DOWN	
	LINE	?	?	θ				LINE	?	?	θ				
Ex. 4	CCW	?	?	?	i	j	UP	CCW	x_1	y_1		i	j	UP	
	LINE	?	?	θ				LINE	?	?	θ				

In the above, modifier P has the following meaning:

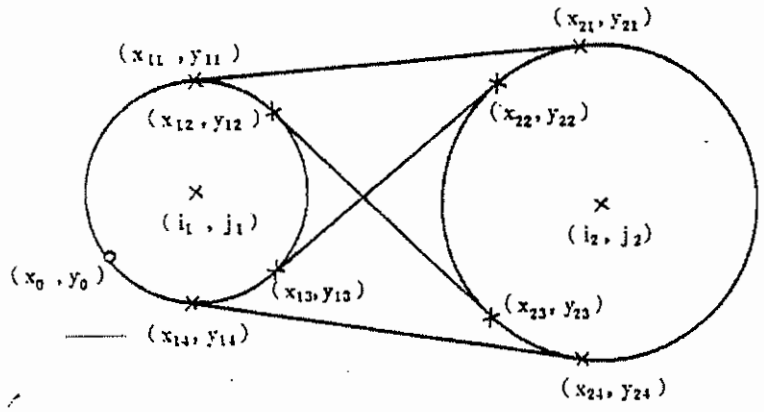
P = ⌐→ : skipped with the cursor.

P = UP : UP specified

P =DOWN: DOWN specified

f. Q-4,Q-3

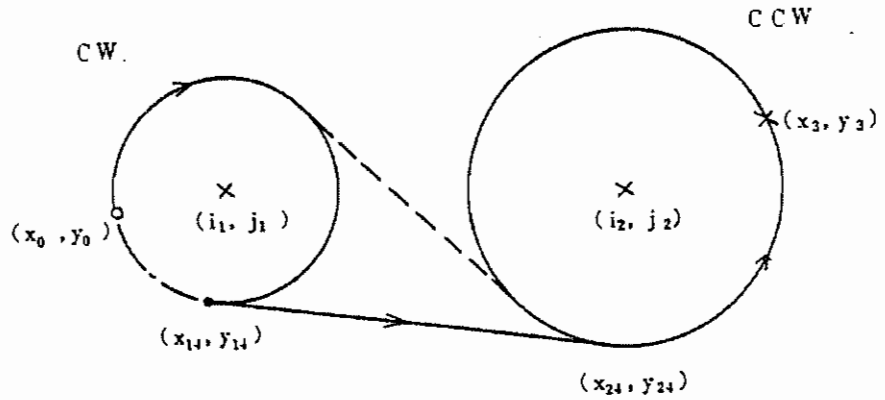
Q-3, however, is definable only when it subsequently comes in contact with an arc. And R-2(or R-3, R-4 or R-5) is required.



Unless modifier P is specified in either block Q-4 or Q-3 the path will be automatically determined as shown in a table below.

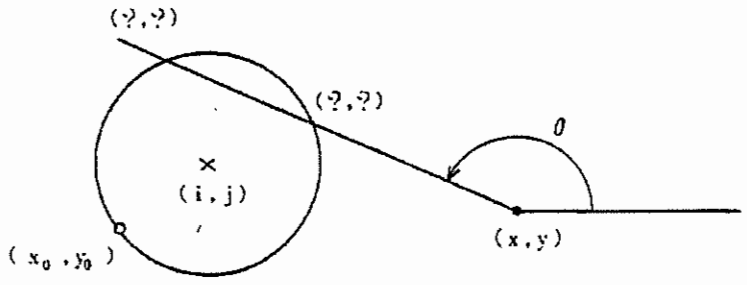
Corner R		Path of Q-3
Q-4	R-2 (or R-3, R-4 or R-5)	
CW	CW	$(X_{11}, Y_{11}) \rightarrow (X_{21}, Y_{21})$
CW	CCW	$(X_{12}, Y_{12}) \rightarrow (X_{23}, Y_{23})$
CCW	CW	$(X_{13}, Y_{13}) \rightarrow (X_{22}, Y_{22})$
CCW	CCW	$(X_{14}, Y_{14}) \rightarrow (X_{24}, Y_{24})$

With mocifier P specified, á contacting point is selected.



Example of Input							Path Selected						
PTN	X	Y	R/θ	I	J	P	PTN	X	Y	R/θ	I	J	P
CW	?	?	?	i_1	j_1	DOWN	CW	X_{14}	Y_{14}		i_1	j_1	DOWN
LINE	?	?	?	?	?	—	LINE	X_{14}	Y_{24}				
CCW	X_3	Y_3		i_2	j_2	°	CCW	X_3	Y_3		i_2	j_2	

g. Q-4, R-1 (or R-6)



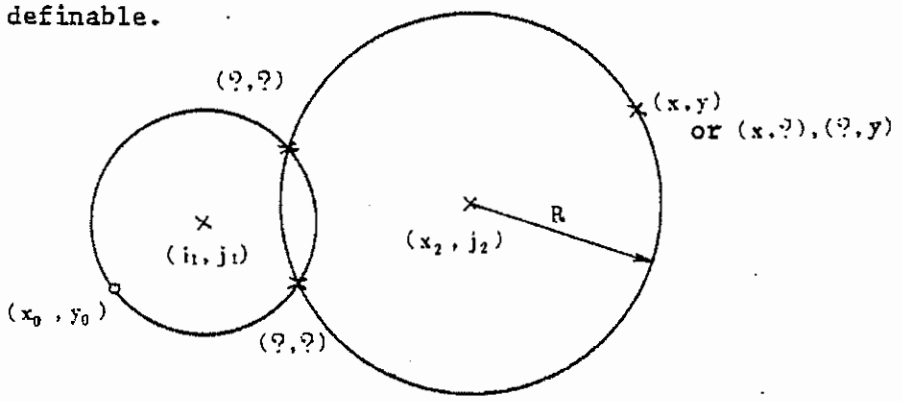
With modifier P specified, right or left crossing point is selected (definable even if broken either to the right or left.)

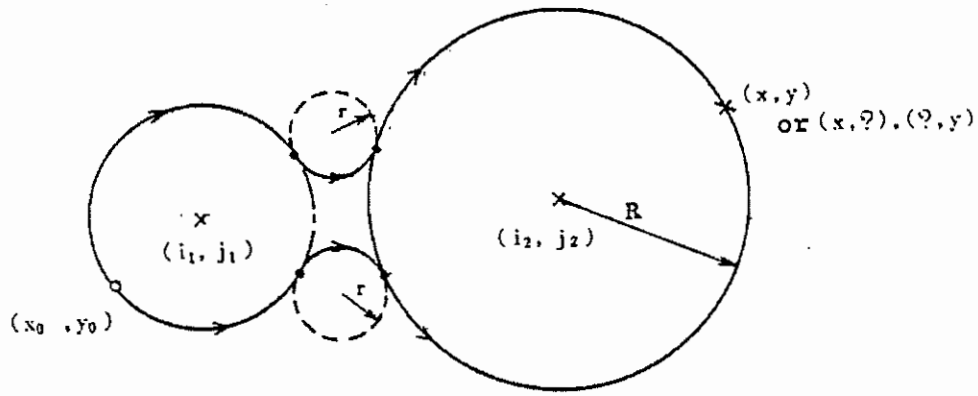
h. Q-4, R-2 (or R3,R4 or R5)

With modifier P specified, a crossing point is selected.

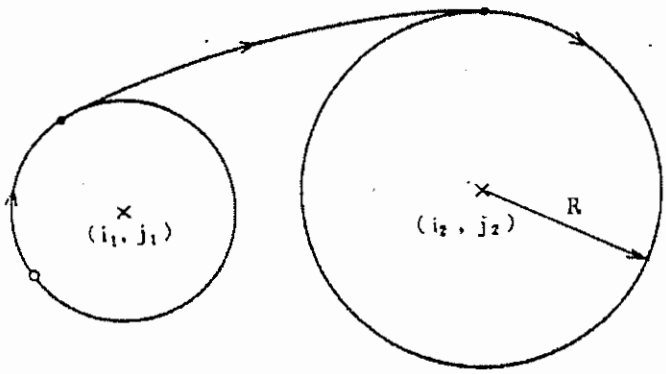
Unless any crossing exists, this combination is undefinable.

With an appropriate corner R inserted, however, it will be definable.





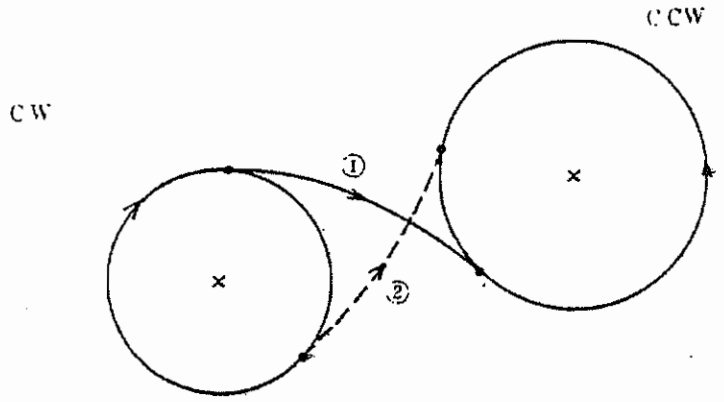
If corner R is larger than radius of both circles, an arc internally touching them should be obtained and reckoned as the corner. (In case where both circles are turning in the same direction.)



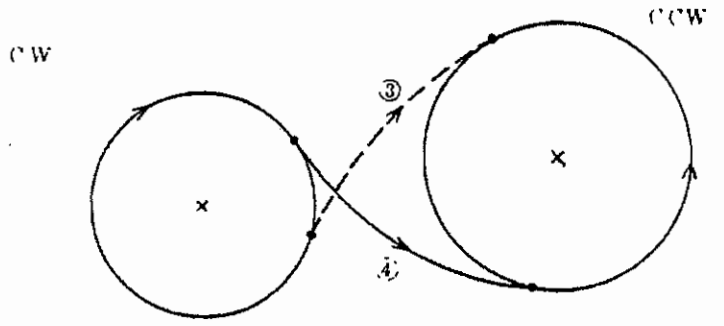
If two detached circle are turning in opposite directions, an arc internally touching the one and externally touching the other is to be reckoned as corner R. In this case, the corner R should be marked to specify positive and negative for CW and CCW, respectively.

The case where Q-4 is CW is shown as an example. (The path will be as shown in a solid line)

With corner R > 0 (CW)



With corner R < 0 (CCW)



A table below shows the relation between turning direction and path.

Turning Direction		Path	
Q - 4	R-2 (or R-3, R-4 or R-5)	with r > 0	with r < 0
CW	CCW	①	④
CCW	CW	③	②

2.1.5 Face machining mode unit :

Depress mode menu [FACE/MACH-ING]

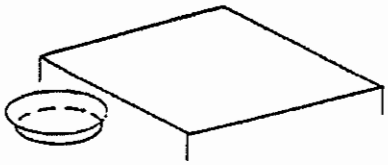
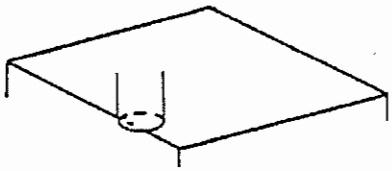
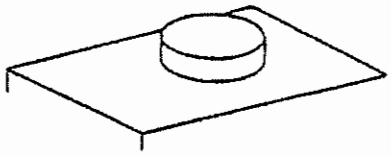
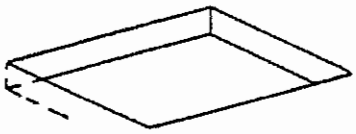
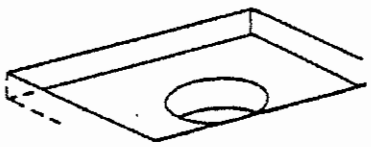
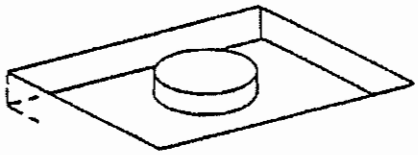
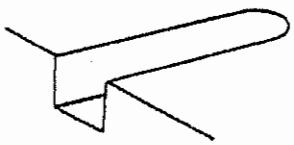
POINT MACH-ING	LINE MACH-ING	FACE MACH-ING	MANUAL PROGRAM	OTHER	WPC	OFFSET	END	GROUP CHECK

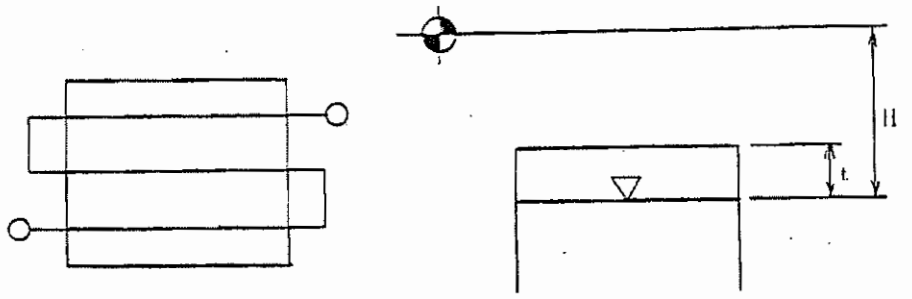
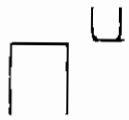
Once the display has changed over to the machining unit selection menu, select a machining unit.

(1) Unit data

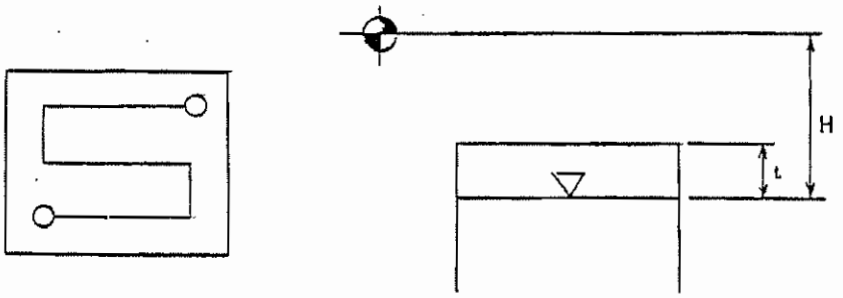
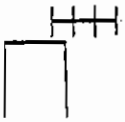
FACE-MIL	TOP EMIL	STEP	POCKET	PCKT MT	PCKT VLY	SLOT	3-D SURFACE (Note)	

Note: The 3-D (dieing) is an optional item of MAZATROL CAM M-2.

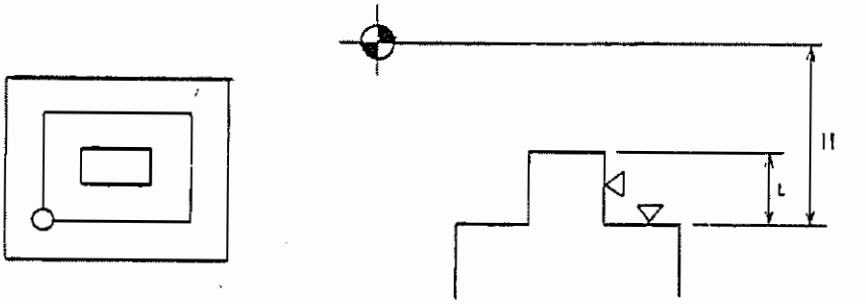
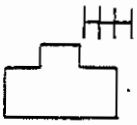
Face milling 	End milling, top 	End milling, mountain 
Pocket end Milling 	Pocket end milling, valley 	Pocket end milling, mountain 
End milling, groove 		

Unit	FACE-MIL	
Menu Key	FACE-MIL 	

Unit	DEPTH (H)	SRV-Z(t)	SRV-R	BTM(▽)	FIN-Z	FIN-R
FACE MIL (M)	999.999	99.999	◆	9 (M)	99.999 (AUTO)	◆

Unit	TOP EMIL	
Menu Key	TOP EMIL 	

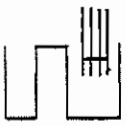
Unit	DEPTH (H)	SRV-Z(t)	SRV-R	BTM(▽)	FIN-Z	FIN-R
TOP EMIL (M)	999.99	99.999	◆	9 (M)	99.999 (AUTO)	◆

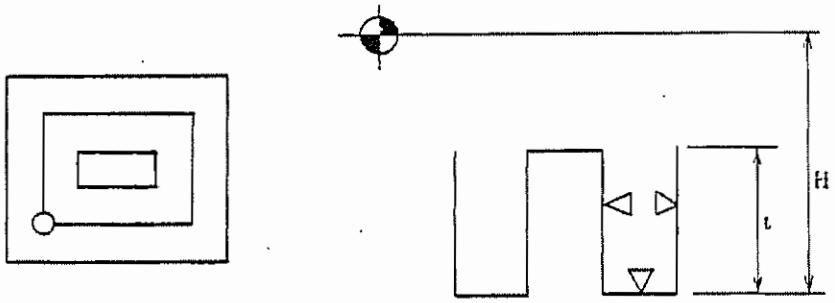
Unit	STEP	
Menu Key	STEP 	

Unit	DEPTH (H)	SRV-Z(t)	SRV-R	BTM(▽)	WAL■(▽)	FIN-Z	FIN-R
STEP (M)	999.999	99.999	◆	9 (M)	9 (M)	99.999 (AUTO)	99.999 (AUTO)

Unit	POCKET
Menu Key	POCKET 

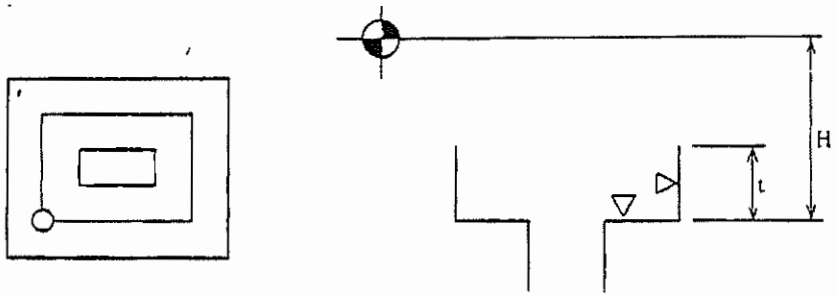
Unit	DEPTH (H)	SRV-Z (t)	SRV-R	BTM (▽)	WAL (▽)	FIN-Z	FIN-R
POCKET (M)	999.999	99.999	◆	9 (M)	9 (M)	99.999 (AUTO)	99.999 (AUTO)

Unit	POKT MT
Menu Key	PCKT MT 



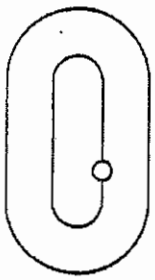
Unit	DEPTH(H)	SRV-Z(t)	SRV-R	BTM(▽)	WAL (▽)	FIN-Z	FIN-R
PCKT MT (M)	999.999	99.999	◆	9 (M)	9 (M)	99.999 (AUTO)	99.999 (AUTO)

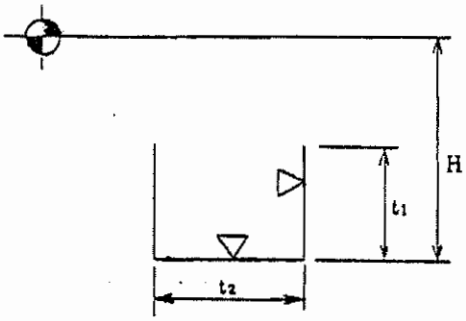
UNIT	PCKT VLY
Menu Key	PCKT VLY 



UNIT	DEPTH(H)	SRV-Z(t)	SRV-R	BTM(▽)	WAL (▽)	FIN-Z	FIN-R
PCKT VLY (M)	999.999	99.999	◆	9 (M)	9 (M)	99.999 (AUTO)	99.999 (AUTO)

UNIT	Groove End Milling	
Menu Key	SLOT	





Unit	DEPTH(H)	SRV-Z(t_1)	SLOT-WID	BTM(▽)WAL(▽)		FIN-Z	FIN-R
SLOT (M)	999.999	99.999	99.999	9 (M)	9 (M)	99.999 (AUTO)	99.999 (AUTO)

(2) Tool Sequence Data

End mill, face mill, chamfering cutter and ball end mill

SNO	TOOL	NOM-ϕ	NO	APRCH-X	APRCH-Y	TYPE	ZFD	DEP-Z	WID-R	C-SP	FR	M	N
R1 (F1)	①	②	③	④	⑤	⑥	⑦	⑧	⑨	⑩	⑪	⑫	⑫

① Tool name (automatically displayed) [TOOL]

ENDMILL	FACEMIL	CHAME. CUTTER	BALL ENDMILL					

② Nominal tool diameter (NOM-ϕ)

It is set with a numerical key. Tools of the same kind are distinguished by depressing the following menu keys.

A	B	C	D	E	F	G	H	J

If the tools have not been registered in the tool file, an alarm will result when the menu key is depressed.

③ Tool priority machining Nos. [NO.]

Proceed to input of tool priority machining numbers (prior machining No./subsequent machining No.).
For input of machining numbers, menu keys and/or ten keys are used:

(1) Prior machining No.
It is input with ten keys only. (1-63)

(2) Subsequent machining No.
The [DELAY PRIORITY] menu key is depressed to inverse the menu. Then, the subsequent machining No. is entered with ten keys. (1-63)

(3) None (Input is unnecessary.)

Menu

	DELAY PRIORITY		PRI. NO. CHNGE	PRI. NO ASSIGN		PRI. NO ALL ERAS	SUB PRO= PROC END	

For details, see "2.2 Same Tool Priority Machining Function".

④ Tool approach point coordinate X [APRCH-X]

⑤ Tool approach point coordinate Y [APRCH-Y]

They are set with numerical keys or with the menu key [AUTO SET]. (④ and ⑤ in common)

⑥ Tool moving methods [TYPE]

Tools are selected with menu keys. (The menu differs with each kind of tool.)

Face mill:

X BI-DIR	Y BI-DIR	X UNI-DIR	Y UNI-DIR	X BI-DIR SHORT	Y BI-DIR SHORT			

End mill:

X BI-DIR	Y BI-DIR	X UNI-DIR	Y UNI-DIR					

Other units:

CW CUT 	CCW CUT 							

⑦ Selection of feed in Z-axis direction [ZFD]

The feed speed at which the tool cuts from the approach point in the Z-axis direction is set in terms of a multiple of feed speed in the radius direction, using ten keys.

Otherwise, either G01 feed (30% of the radius direction feed rate) (see Note) or G00 feed (rapid feed) is selected with menu keys.

JOG G01	RAPID G00							

Note: Thirty percent is the standard value set by means of line and face machining parameters.

⑧ Penetration in an axial direction in one operation [DEP-Z]

⑨ Penetration in radial direction in one operation [WID-R]

AUTO SET

This is automatically calculated when the AUTO SET menu key is pressed. [WID-R]

The numerical keys will allow input or change of value.

⑩ Peripheral speed [C-SP]

⑪ Feed [FR]

HSS AUTO	CARBIDE AUTO

When the tool tip material is selected with the menu key, the peripheral speed (/min) and feed speed (/rev.) will be automatically determined.

Note : For ball end mill, it is not automatically determined.

⑫ M code [M]

Data can be inputted either by selecting with a menu key or by using ten keys. (The former M can be inputted in two digits while the latter can be inputted in three digits.)

2.1.6 MANUAL PROGRAM mode unit

Depress the MANUAL PROGRAM in the mode menu.

POINT MACH-ING	LINE MACH-ING	FACE MACH-ING	MANUAL PROGRAM	OTHER	WPC	OFFSET	END	GROUP CHECK

The manual program unit is used to execute a conventional program. One manual program mode unit involves only one tool.

(1) Unit data

UNO	UNIT	TOOL	NOM-ϕ	NO
	MANU PRO (M)	①		②

① Tool name [TOOL]

Menu (I)

ENDMILL	FACEMILL	CHAMF. CUTTER	BALL END MILL	OTHER TOOL	TOUCH SENSOR		NO TOOL	NEXT (1/2)

Menu (II)

CENTER DRILL	DRILL.	BACKSPOT FACER	REAMER	TAP	BORING BAR	BACK BOR.BAR	CHIP VACUUM	NEXT (2/2)

Menu NO TOOL is selected, it operates as if the tool number goes 0.

② Tool priority machining Nos. [NO.]

For input of priority machining numbers, menu keys and/or ten keys are used. (See "2.2 Same Tool Priority Machining Function").

(2) Sequence data

SNO	G1	G2	DATA1	DATA2	DATA3	DATA4	DATA5	DATA6	S	M/B
1	①	①	②	②	②	②	②	②	③	④
⋮										

① G code

Two codes can be inputted per one line of sequence. They are set with menu keys or numerical keys.

G00	G01	G02	G03	G40 CANCEL	G41 LEFT	G42 RIGHT	MANUAL END	

G codes usable:		(Note 1)	Group(Note 3)
G00	Positioning (quick feed)	Modal	Ⓐ
G01	Linear interpolation	Modal	A
G02	Circular arc interpolation (Clockwise: CW)	Modal	A
G03	Circular arc interpolation (Counterclockwise: CCW)	Modal (Note 2)	A
G04	Dwell	Non-modal	A'
G17	Plane selection (Selection of XY plane)	Modal	Ⓑ
G18	Plane selection (Selection of ZX plane)	Modal	B
G19	Plane selection (Selection of YZ plane)	Modal	B
G28	Return to original point I	Non-modal	A"
G30	Return to original point II	Non-modal	A"
G40	Tool diameter correction cancel	Modal	Ⓒ
G41	Tool diameter correction left	Modal	C
G42	Tool diameter correction right	Modal	C
G90	Absolute command	Modal	Ⓓ
G91	Incremental command	Modal	D
G94	Asychronous feed command	Modal	E
G95	Synchronous feed command	Modal	Ⓔ

Note 1. The modal G code is a G code which remains valid until another G code in the same group is commanded.
The group is a gathering of the modal G codes which are interrelated one another. There are groups, A through E, according to the function given to each of the groups.

2. The unmodal G code is a G code which is valid only for that particular block.

3. The G code marked with ◯ is a modal G code which is automatically selected when the manual program mode starts. When one of them is used at the top of the program, it is not necessary to specify it.

② Various kinds of data

A maximum of six data can be put in one sequence line.
Data to be entered are inputted with the menu key and data values, with numerical keys.

Menu (I)

X	Y	Z	4	F	R		DATA CANCEL	NEXT (1/2)

Menu (II)

I	J	K	PITCH (P)	DWELL (D)			DATA CANCEL	NEXT (2/2)

Each time the NEXT is depressed, menu (I) and menu (II) appear on the screen alternately.

Data	Purpose		Effective command value	Remarks
X	Coordinate system position data		0 ∿ ±9999.999 [mm]	
Y	Coordinate system position data		0 ∿ ±9999.999 [mm]	
Z	Coordinate system position data		0 ∿ ±9999.999 [mm]	
4	Additional axis coordinate system position data		0 ∿ ±9999.999 [mm]	
F	Feed speed	Asynchronous feed	0 ∿ 9999 [mm/min]	Integer command
		Synchronous feed	0 ∿ 999.999 [mm/rev.]	
R	Arc whose radius R is specified		0 ∿ ±9999.999 [mm]	
I	Incremental stroke from the starting point to the coordinates of the arc center		0 ∿ ±9999.999 [mm]	
J	Incremental stroke from the starting point to the coordinates of the arc center		0 ∿ ±9999.999	
K	Incremental stroke from the starting point to the coordinates of the arc center		0 ∿ ±9999.999 [mm]	

Data	Purpose	Effective command value	Remarks
P	Helical cut pitch	0 ∿ ±9999.999 [mm]	Depress the PITCH menu key
D	Dowelling time data	0 ∿ 999.999 [SEC]	Depress the DOWEL menu key

③ S code

Spindle function code (5 digits, 0 ∿ 65535)

④ M/B code

M: Auxiliary function code (0 ∿ 999)

B: Rotary table auxiliary function code (0 ∿ 999)

Note 1: For input of the B code, the B CODE INPUT must be depressed.

Note 2: Mirror image M90, M91 and M92 commands should be given in the M code unit. They will be unavailable even if given in the single action unit (MANU PRO).

2.1.7 Specialmode unit

Depress the OTHER mode menu key.

POINT MACH-ING	LINE MACH-ING	FACE MACH-ING	MANUAL PROGRAM	OTHER	WPC	OFFSET	END	GROUP CHECK

Select the unit with the menu:

M CODE	SUB PROGRAM	MMS	DRUM CHANGE	PALL CHG	INDEX	PROCESS END		

(i) M code unit

When the mode menu key M CODE is depressed, the following appears on the screen:

UNO UNIT NO M1 M2 M3 M4 M5 M6 M7 M8
M CODE ①
(M)

① Tool priority machining Nos. [NO.]

For input of priority machining numbers, menu keys and/or ten keys are used. (See "2.2 Same Tool Priority Machining Function").

(ii) Sub-Program unit

When the mode menu key [SUB PROGRAM] is pushed, the screen displays the following:

UNO	UNIT SUB PRO	WNO.	REPEAT	ARGM1	ARGM2	ARGM3	ARGM4	ARGM5

The definition of this unit allows calling and execution of other programs. In case the same action is repeated many times or the same action is used by a plural number of programs, a program concerning such action may be specially developed and called by this unit. The program thus calling other programs is referred to as the main program while the programs thus called are referred to as sub-programs. Calling a sub-program is referred to as nesting.

Both MAZATROL Program and EIA/ISO Program can nest sub-programs. However, the MAZATROL Program can nest sub-programs nine times at the most while the EIA/ISO Program can call them eight times at the most.

WORK NO.: Input the WORK NO. of the sub-program to be executed.

REPEAT : Input the number of repetitions of the specified sub-program.

ARGM : Input the ARGM applicable to calling of the user macro (option).

Note 1: Work Nos. are classified according to two levels:
9000's: Edition is impossible when a background program is being prepared.
Others: Edition is possible even when a background program is being prepared.

Note 2: The auxiliary coordinates specified by the main program are valid for subprograms. However, auxiliary coordinates specified by a subprogram are valid only for that sub-program. When the operation has returned to the main program, the auxiliary coordinates which were valid before

calling of the subprogram again control.

If fundamental coordinates are established in a subprogram, the auxiliary coordinates in the main program are cancelled.

Note 3: Always enter a number in NUMBER.

(iii) MMS Unit

See "MAZAK Machining Monitor System" which will be separately furnished.

(iv) Drum change unit (for machines equipped with drum changers)

Depressing the mode menu key [DRUM CHANGE] gives the following display on the picture:

[UNO]	UNIT	DRUM NO.
	DRUM CHG	

Enter the drum No. (1-4) to be called.

Note 1: Input of this unit defines the effective range of tool priority machining numbers. (See "2.2.5 Range of tool priority machining (process)").

(v) Pallet change unit

When the mode menu key [PALL CHG] is pushed, the following display is given:

[UNO]	UNIT	PALLET NO
	PALL CHG	

Input data for the pallet change unit include the command pallet No. and next pallet No. (entered within the parentheses).

Command pallet No.: Pallet No. to be replaced by means of the pallet change unit.

Next pallet No.: Pallet No. to be replaced by means of the next pallet change unit
(Input is possible only when the machine is provided with the next pallet change system.)

Note 1 : Input of this unit determines the effective range of tool priority machining Nos. When a pallet is to be changed, it is necessary to use the pallet change unit in principle.
(See "2.2.5 Range of tool priority machining (process)".)

Note 2 : The pallet No. currently present on the table is displayed on the POSITION and COMMAND display. This pallet No. is updated when pallet change is executed by the pallet change unit but it is not updated when the pallet change is executed by an M code or manually. In the latter cases, change the pallet No. in the MDI mode.
In case operation has been suspended by resetting, do not forget to set a correct pallet No. value.

Note 3 : Pallet change hold switch and pallet number display.

Program	
	Pallet change 1
	Pallet change 0
	End

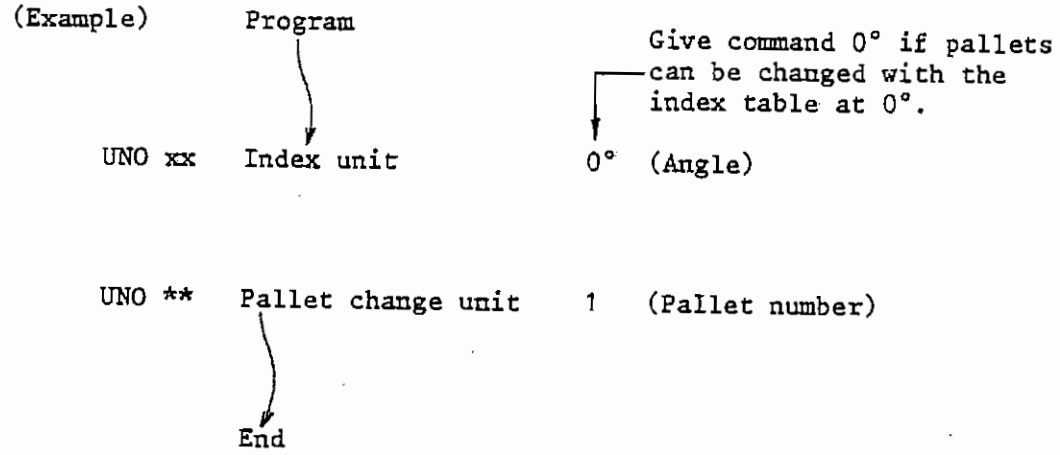
In the above program, turn on the pallet change hold switch after transferring a pallet from the table to the pallet stand using pallet change (PALL CHG) 1. Then, after the axis has moved, the pallet will not change and no pallet will exist on the table. However, the CRT displays pallet No. "1". This indicates the number of the pallet to be placed on the table after resetting the pallet change hold mode. If the NC unit has been reset under the conditions mentioned above, a pallet number must be input again.
If the system is in the pallet change hold mode at the start of the pallet change (PALL CHG) 0 operation, the CRT will display pallet No. "0". Likewise, there will be no pallet on the table after resetting the pallet change hold mode.

Note 4 : In the H15J provided with 2PC, pallet No. "0" cannot be used. In programming, always use pallet No. 1 or 2. When pallet No. 0 is displayed on the CRT, input pallet No. 1 or 2 using PALL NO SET menu.

Note 5 : When there is a stacker crane, enter 2 in the current pallet number (PALLET NO.) on the CRT. This number cannot be updated.
Note: Pallet 2 command does not cause the system to operate. If pallet is to operate, give the command, using an M or G code.

Note 6 : To execute the pallet change unit (PALL CHG) in a machine equipped with an index table, the index angle must be set in the range of angles in which pallets can be changed by providing an index unit before the pallet change unit.

(Example)



Note 7 : The use of this unit requires the response by the PC sequence. Be careful in adopting a new version of NC (E models and upwards).

(vi) Index unit

This unit is used to machine each plane indexed in the index table with the same tool, without changing the tool.
With the same tool priority function available on every plane, this unit is required.
When the mode menu [INDEX] is depressed, the following display is given:

[UNO]	UNIT	TURN POS X	TURN POS Y	TURN POS Z	ANGLE	TURN DIR
	INDEX					

The definition of this unit allows the control of the angle of the machined surface.
When tools (tools for prior, regular and subsequent machining) are searched in a machine to which the index table and NC rotary tables are applicable, the tool is changed after execution of the last index unit found during search. Also, when the pallet change, drum change or process end unit is searched, the unit is executed after executing the last index unit found during search.

TURN POS: Swing position coordinates is inputted (entered using the machine coordinate system).
No axis will be moved unless inputted.

ANGLE : The table angle data is entered by "absolute" in either case of the NC rotary table, B code index table and M code index table.

TURN DIR: Display the following menu and set the swing direction with a menu key. (Applicable to the B code index table only.)

NEAR DIR (AUTO)	CW	CCW						

Note 1 : B code index table

The NC regards as "0" the table angle when starting a program. Thereafter, the index position memory is updated whenever each index unit is executed.

However, the memory is not updated when indexing is executed by other means (by MDI or manually) than the index unit.

When the index table does not specify "0°" upon starting a program or when indexing is executed by other means than using the index unit, the direction may be reversed if indexing is executed by a short-cut indexing method.

The direction may also be reversed when resetting it executed during indexing. Always set the table angle to 0° when starting a program.

Upon restart, set the table angle in advance to the angle at the restarting point. In this case, if the second tool has a priority number the table will not swing but the axis will only move to the indexed swing position when the plane to be machined at the angle which is the same as the first one after restart.

Note 2 : M code index table

Because the NC regards as "0°" the table angle when starting a program, set the table angle as "0°" by the MDI or manually if the table angle is not "0°".

If the index unit is then executed, the determined M code (AF2: MACH CONSTANT PAR NO. 2) is output as often as the frequency, where the table reaches the specified positions. Such frequency is determined by the minimum indexing angle (parameter: AF1).

Example: AF1: 90 and AF2 = 45

Head of program:—The NC regards as "0" the table angle upon starting program.

Index 90°: ← $\frac{\text{Command value - Angle at present}}{\text{Minimum indexing angle}}$ =

$\frac{90° - 0°}{90°} = 1$

Therefore, M45 is output once. After execution, the internal memory stores 90°.

Index 270°: ← $\frac{270° - 90°}{90°} = 2$

Therefore, M45 is output twice. The indexing is clamped at the 180° position and then the index is set at the 270° position.

Index 0°: ← $\frac{0° - 270° + 360°}{90°} = 1$ (360° is added when command value is greater than the current value.)

Therefore, M45 is output once.

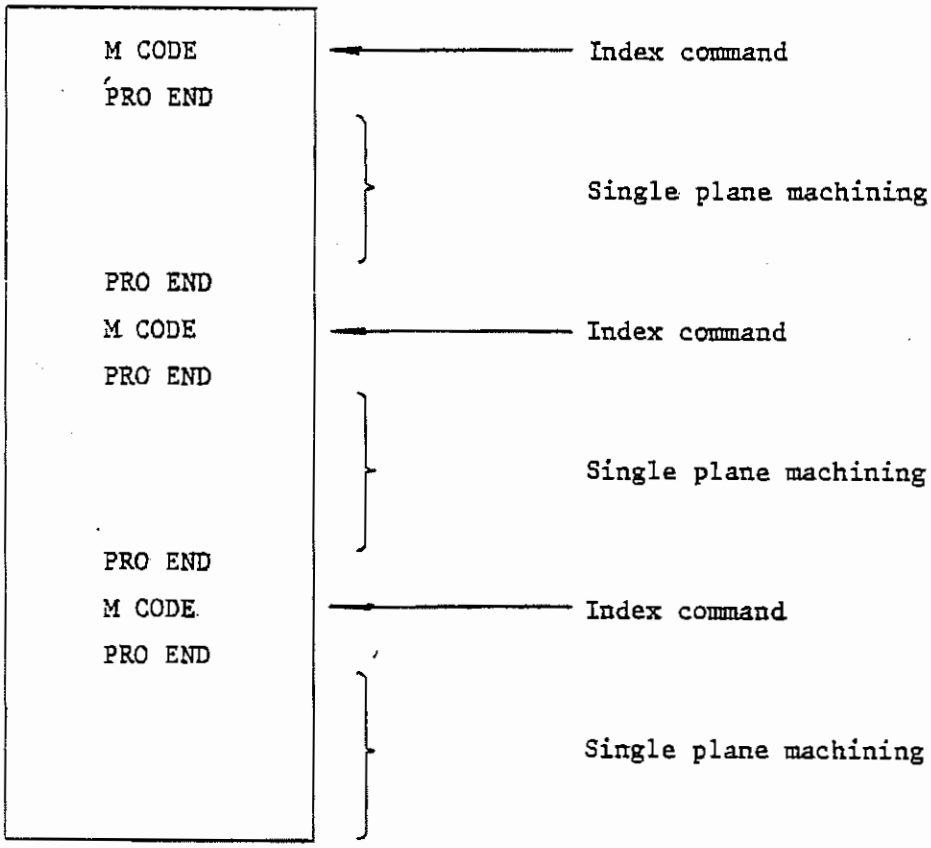
When indexing is executed using an M code in the MDI or manually, the index position memory inside the NC is not updated. In principle, the M code index table should not be used for indexing by means of MDI or manual interruption.

If it is absolutely necessary to do so, index by MDI or manually and then execute the start after restoring the index angle to the value valid before interruption.

In case operation has been suspended by resetting, the data stored in the position memory inside of the NC differs from the current value. Therefore, set the index as "0°" by MDI or manually, switch on and off the power and execute restart.

Restart is possible only on the plane machined at a table angle of 0°.

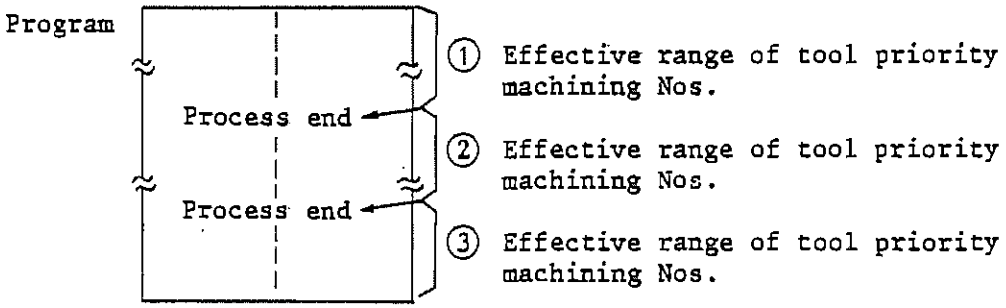
Note 3 : By using the index unit, it is not possible to control the M code index table which requires two or more M codes to carry out a cycle of indexing or such index table as programs are executed at every step with an M code from the NC unit, with a programmable controller on the index side.
Use the conventional M code unit to give commands.
In this case, moreover, it is not possible to use the same tool priority function over a wide diversity of angles. With the process end unit (PRO END) used to delimit before and after the index M code unit, use the same tool priority function taking such a delimited angle as a range of the same tool priority (one process).



(vii) Process end unit

This unit specifies the effective range of the tool priority machining Nos. When the mode menu [PROCESS END] is depressed, the following display is given:

```
[UNO]        UNIT
             PRO END
```



NOTE 1: The effective range of the tool priority machining Nos. can be also defined by the pallet change unit. (See "2.2.5 Range of tool priority machining (process)").

2.1.8 Auxiliary coordinate system unit

Depress the [OFS] mode menu key.

POINT MACH-ING	LINE MACH-ING	FACE MACH-ING	MANUAL PROGRAM	OTHER	WPC	OFFSET	END	GROUP CHECK

The following display appears on the screen.

UNO	UNIT OFFSET (M)	U(X)	V(Y)	D(θ)	W(Z)

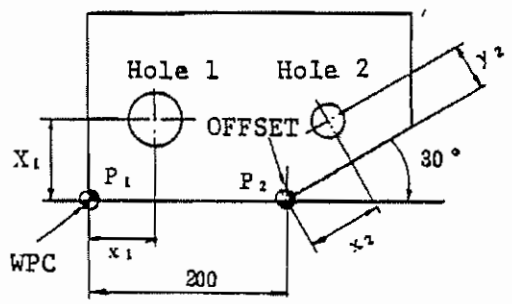
① The auxiliary coordinate system unit is used to transfer the zero program position (WPC) to any desired position for ease of program execution. (The auxiliary coordinate system unit is used during execution of a program.)

② Contents of data (See the figure below.)

U(X), V(Y), W(Z): Amount of transfer from program zero point (0 to ±9999.999)

D(θ) : Angle formed by work coordinate system and auxiliary coordinate system (0 to ±999.999)

UNO	UNIT	U(X)	V(Y)	D(O)	W(Z)
1	OFFSET	200.000	0	30	0



Dimensions of Hole 1 are given in reference to point P1 while those of Hole 2 are given in reference to point P2.

P1-WPC

P2 can easily be obtained when the program is executed in the OFFSET mode.

Note 1) The auxiliary coordinate specified by a subprogram is cancelled when the control is returned to the main program.

2.1.9 End unit

Depress the [END] mode menu key.

POINT MACH-ING	LINE MACH-ING	FACE MACH-ING	MANUAL PROGRAM	OTHER	WPC	OFFSET	END	GROUP CHECK

The following display will appear on the screen.

UNO	UNIT	CONTI	NUMBER
	END (M)		

① This unit is set at the end of the program.
The count command can be set in the end unit.

② Contents of data

CONTINUE: Indicates whether the program should be executed repeatedly or not. (executed: 1, not executed: 0)

NUMBER : Each time this program has been executed, its frequency of execution is indicated by the parts counter on the position screen.
(Counted: 1, Not counted: 0)

Note 1: The count set value can be set using the position command screen. The cumulative count value can be also cleared.

Note 2: This unit cannot be entered midway through a program.

2.2 Same Tool Priority Machining Function

2.2.1 Description of function

The present function gives numbers to automatically developed tools in the order of machining to reduce the number of ATC's and shorten the machining time by executing machining in such an order.

2.2.2 Input points

① Tool sequence data for point/line/surface machining units
② Manual program mode unit data (invalid when there is no tool)
③ MMS unit data
④ M code unit data

2.2.3 Procedures for input and editing function

① Input of priority machining Nos.

Priority machining Nos. include the prior machining number and subsequent machining number. They are entered with menu keys and ten keys. Nos. are given to tools so that numbers may become greater by the order of the automatic tool development for tools in each unit.

There are three methods of machining number priority input.

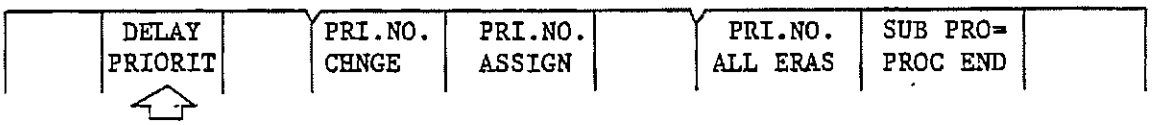
By moving the cursor to the position of "NO" and the number is input in the following manner:

a. Prior machining number: The number is input with ten keys. The number is displayed in white.

b. Subsequent machining number: After depressing the DELAY PRIORITY to inverse the menu, the later machining number is input with ten keys. The number is displayed in white on the pink inversed picture. To erase the inversion (to return to prior machining), depress the DELAY PRIORITY again.

c. No number is specified: Nothing is entered as input. Move the cursor to the next position.

Menu



The order of machining in each case of a, b and c above will be discussed in "2.2.4".

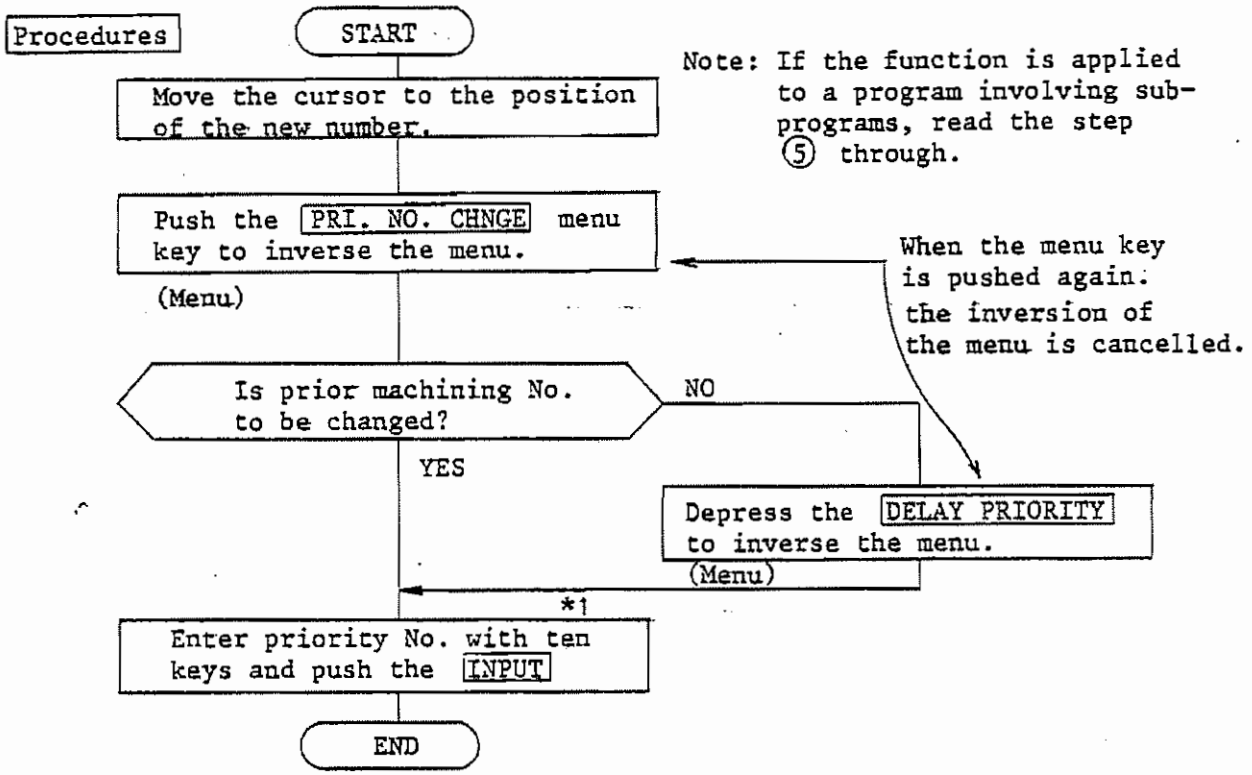
Note 1: For both prior and subsequent machining numbers, 1-63 may be adopted. These numbers may not necessarily serial. (Total; 126)

Note 2: When the same tool is used for a number of purposes, the same priority number may be specified for each purpose. If the same number is allotted to different tools, an alarm will result.
However, it is possible to allot different numbers to the same tool.

Note 3: To cancel a number already entered, put the cursor to the position of that number and push [0 ·] (ten key) and CLEAR .

② Change of priority No.

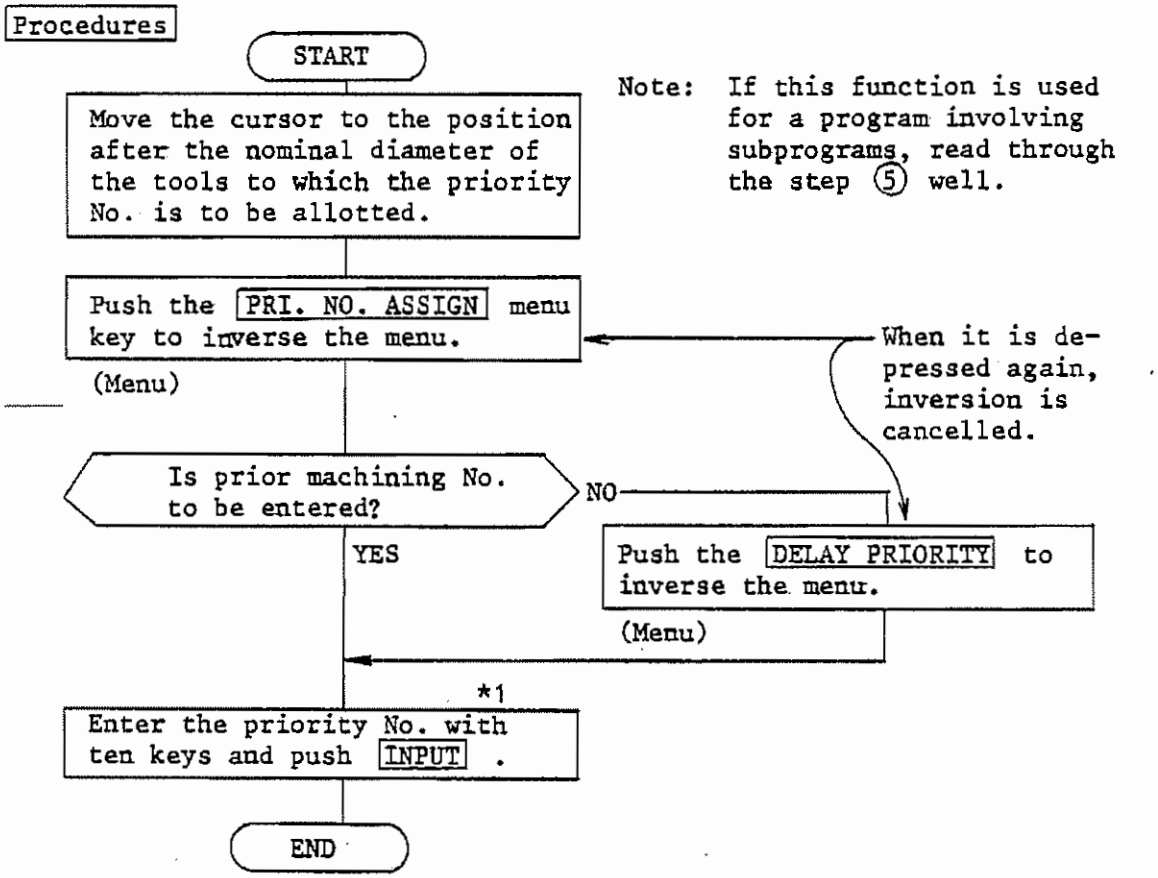
This function is used to renew the same priority numbers used in a process. (See the step ⑤ below.)



To cancel the same priority No. within the same process, push [0.] and [CLEAR] in *1 after practicing the same procedures described above.

③ Allotment of priority numbers

A specified priority number is forcibly allotted to the same kind of tools in the same process (see ⑤) in the following manner:



Note: If this function is used for a program involving subprograms, read through the step ⑤ well.

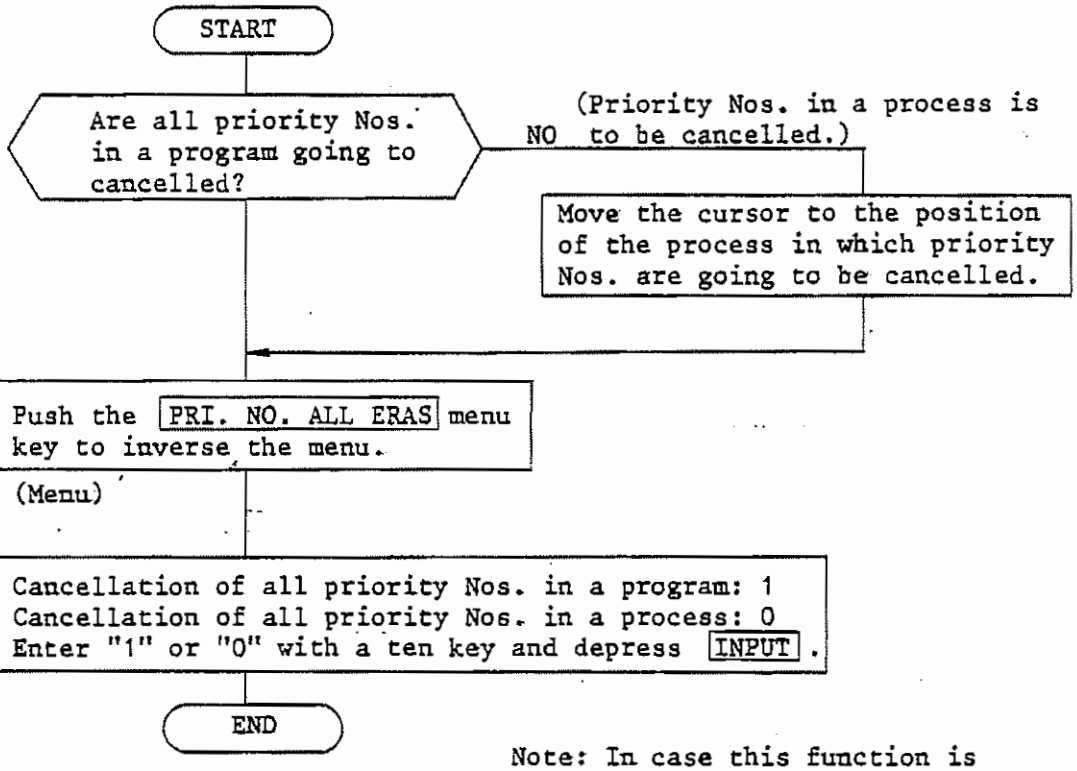
To erase the priority No. for all the tools of the same kind in the same process, push [°.] and [CLEAR] in *1 after practicing the procedures described above.

Note: Be careful because this command is valid for all the tools of the same kind in a process whether or not priority numbers are registered.

④ Cancellation of all priority numbers

All the priority numbers in a process (see ⑤) or in a program are cancelled in the following manner:

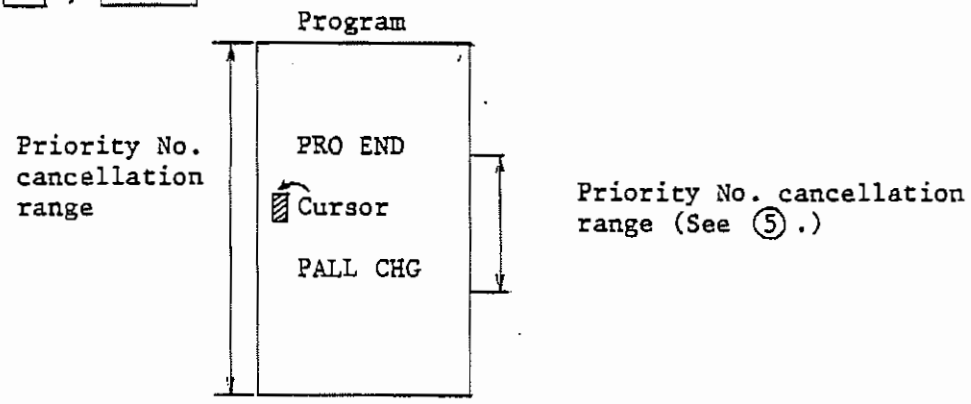
Procedures



Note: In case this function is applied to a program involving subprograms, read through the step ⑤ well.

1 , INPUT

0 , INPUT



⑤ Utilization of process and SUB PRO = PROC END

The effective range of the priority numbers editing function (② change of priority numbers, ③ allotment of priority numbers and ④ cancellation of all priority numbers) and duplicate registration check range are ranges defined by Process end (PRO END), Pallet change (PALL CHG) and Drum change (DRUM CHG) unit.

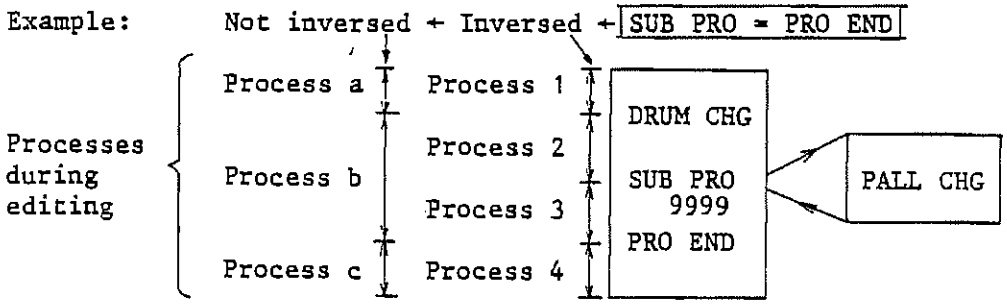
These ranges are the same as the effective range of priority numbers and are called "a process".
(See 2.2.5 Range of tool priority machining.)

When this editing function is applied to a program involving subprograms, the editing is executed only in the main program and no editing is executed in subprograms. If any unit in a subprogram defines the end of a process, it is ignored because search is impossible during editing. For these reasons, practice the following steps if a program involves subprograms:

a. If the priority number of the main program has been edited, edit subprograms in the same manner.

b. If any subprogram having a unit defining the end of a process is present in a process to be edited, depress SUB PRO = PRO END to inverse the menu and then execute editing. Then, the subprogram unit inside of the process undergoing editing is treated like the PRO END and the process for editing is formed.

Example:



2.2.4 Order of machining according to priority numbers

Machining is executed in the following order:

① Machining by tools to which prior machining numbers are allotted is executed in the number order. (Prior machining)

② Machining by tools having no prior or subsequent machining numbers is executed according to the program. (ordinary machining)

③ Machining by tools to which subsequent machining numbers are allotted is executed in the number order. (Subsequent machining)

Note 1: If the same number is given to the tools used at different machining positions, machining is executed at such machining positions according to the program.

Note 2: The subprogram unit during priority search machining is processed in the following manner:

1 If the subprogram is a MAZATROL program, processing is always executed. (Machining included in the subprogram is processed according to priority numbers.)

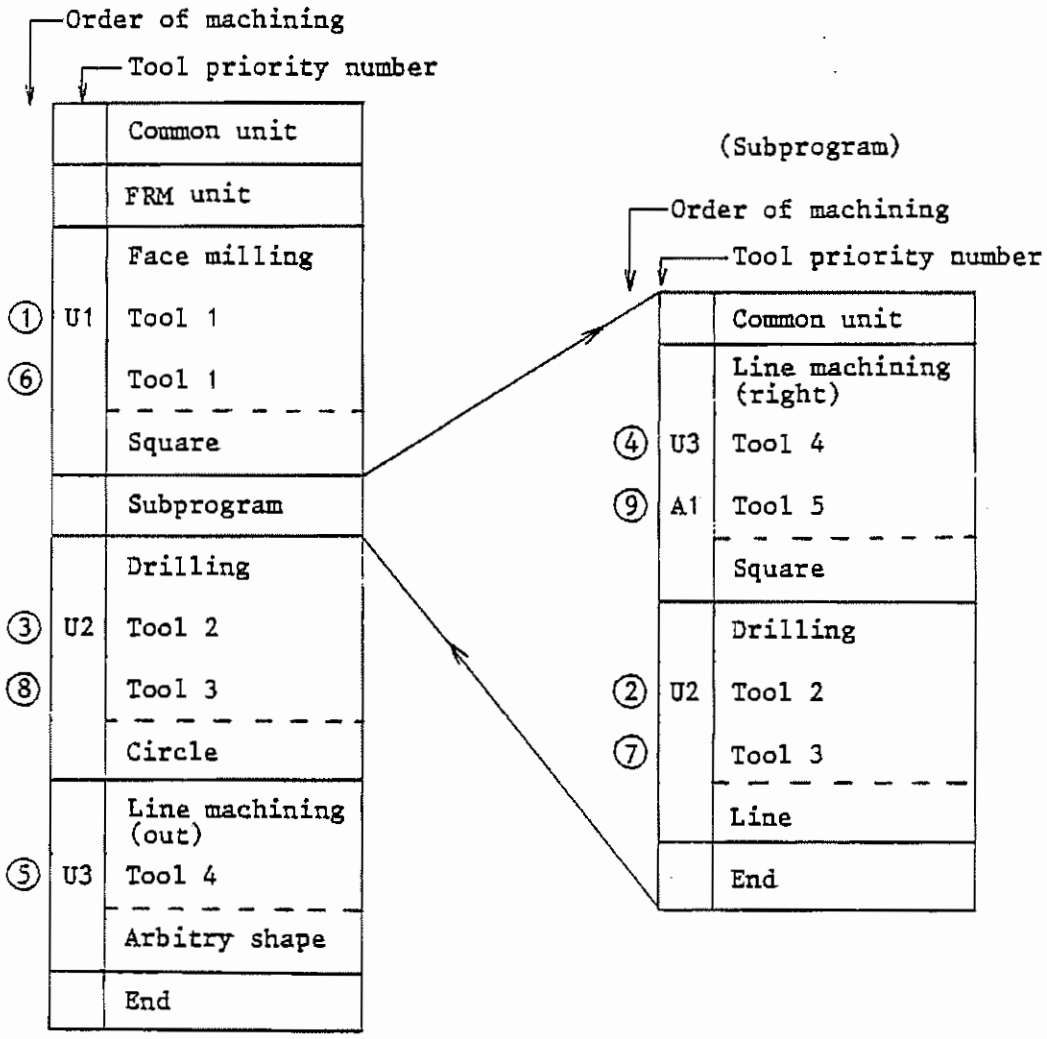
2 If the subprogram unit is EIA/ISO, no processing is executed. Therefore, the EIA/ISO program executes machining in the ordinary machining order.

Note 3: If priority numbers are given by an M code unit, machining is executed only once according to these numbers. If an M code unit does not specify a priority number, machining is executed whenever a priority number is found during priority search.

Note 4: After machining priority search, the index unit read during search made before starting of machining by the tool concerned is executed.

Note 5: After all the tools involved in the machining process have been given priority machining numbers, the M code unit, without a number, will be executed again during the interval between the prior and the subsequent machining.

[Example of machining]



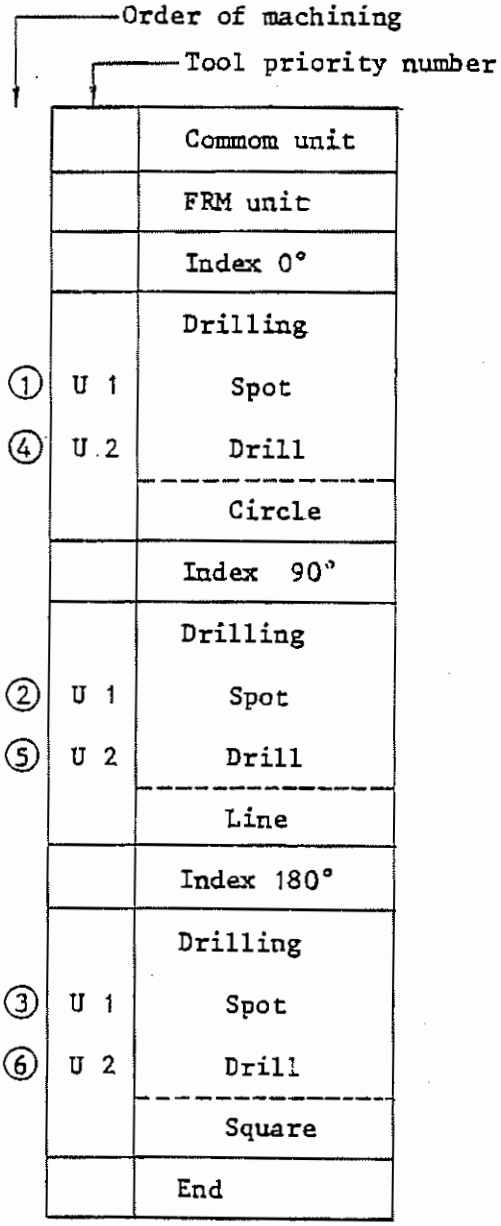
U ..... Prior machining number

A ..... Subsequent machining number

B code and M code index tables or NC rotary table and priority machining

When machining with the table angles being controlled, more effective machining will be possible by using the same tool priority machining function and the index unit (See 2.1.7 "Special mode unit" (vii.) Index unit) together.

(Example)



(Order)

Index 0°
↓
① Spot machining Circle
↓
Index 90°
↓
② Spot machining Line
↓
Index 180°
↓
③ Spot machining Square
↓
Index 0°
↓
④ Drill machining Circle
↓
Index 90°
↓
⑤ Drill machining Line
↓
Index 180°
↓
⑥ Drill machining Square

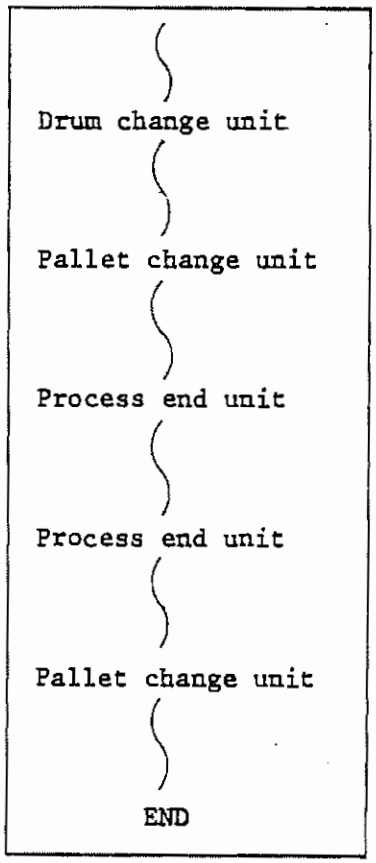
U .... Prior machining number

2.2.5 Range of tool priority machining (process)

■ The following units are used to define the end of a tool priority machining range:

① Drum change unit (See 2.1.6-(iv).)
② Pallet change unit (See 2.1.6-(v).)
③ Process end unit (See 2.1.6-(vii).)

Program



Range of tool priority machining range ① (Called Process ①)

Range of tool priority machining range ② (Called Process ②)

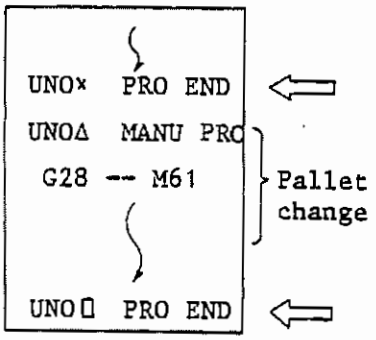
Range of tool priority machining range ③ (Called Process ③)

Range of tool priority machining range ④ (Called Process ④)

Range of tool priority machining range ⑤ (Called Process ⑤)

Range of tool priority machining range ⑥ (Called Process ⑥)

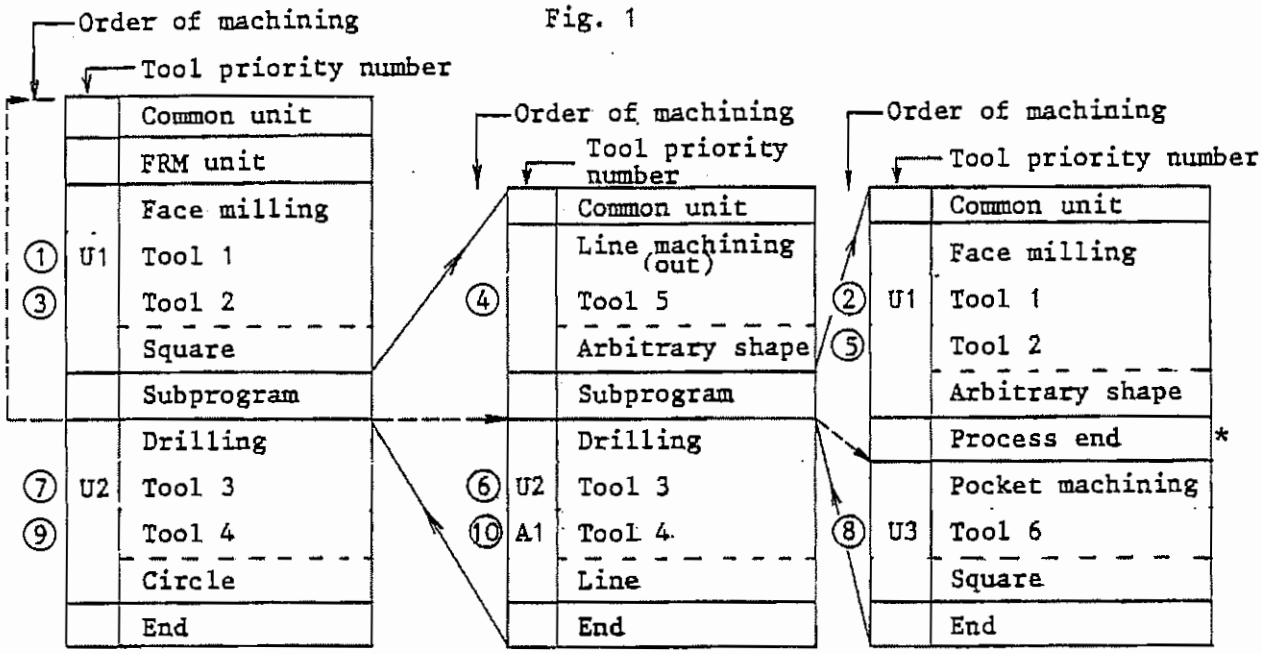
In the above case, machining is executed in the order of tool priority machining processes ① through ⑥. In each process, machining is executed by priority machining number.



Note: For the pallet change function by the manual program mode unit and M codes and the same tool priority function are combined and used in the same program, it is necessary to put the process end unit before and after the manual program mode unit and M codes for the pallet change.

When unit defining end is present in a subprogram:

Fig. 1

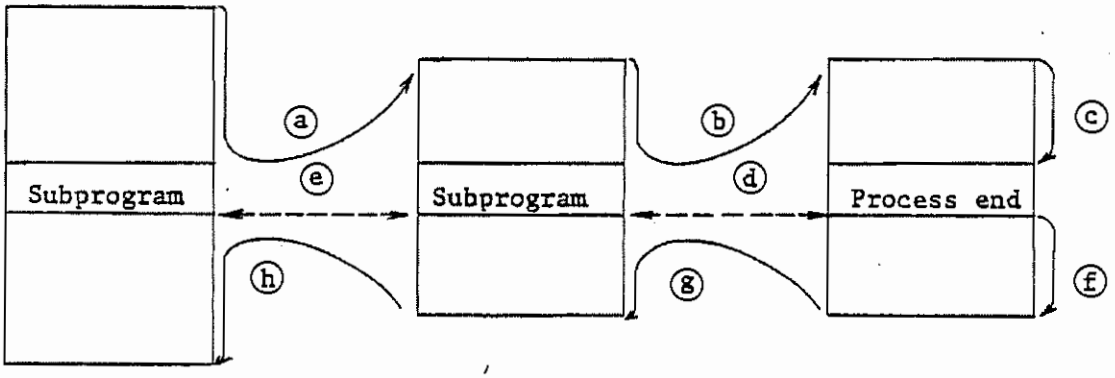


U: Prior machining number

A: Subsequent machining No.

* End of priority machining number
Process end, pallet change, drum change

Fig. 2



If process end unit is present in a subprogram as shown in Fig. 1, the same tool priority function is executed in two ways as seen in Fig. 2:

Starting the execution

Process 1 ⓐ → ⓑ → ⓒ → ⓓ → ⓔ

Process 2 ⓕ → ⓖ → ⓗ → ⓔ → ⓓ

2.2.6 Same tool priority function in multi-piece machining

(1) When nesting of the multi-piece machining function and the same tool priority function are duplicated, machining is executed in the following order:

(Explanation is given in reference with the same program composition composition describing nesting function for multi-piece machining in 2.1.1, to which priority numbers are attached.)

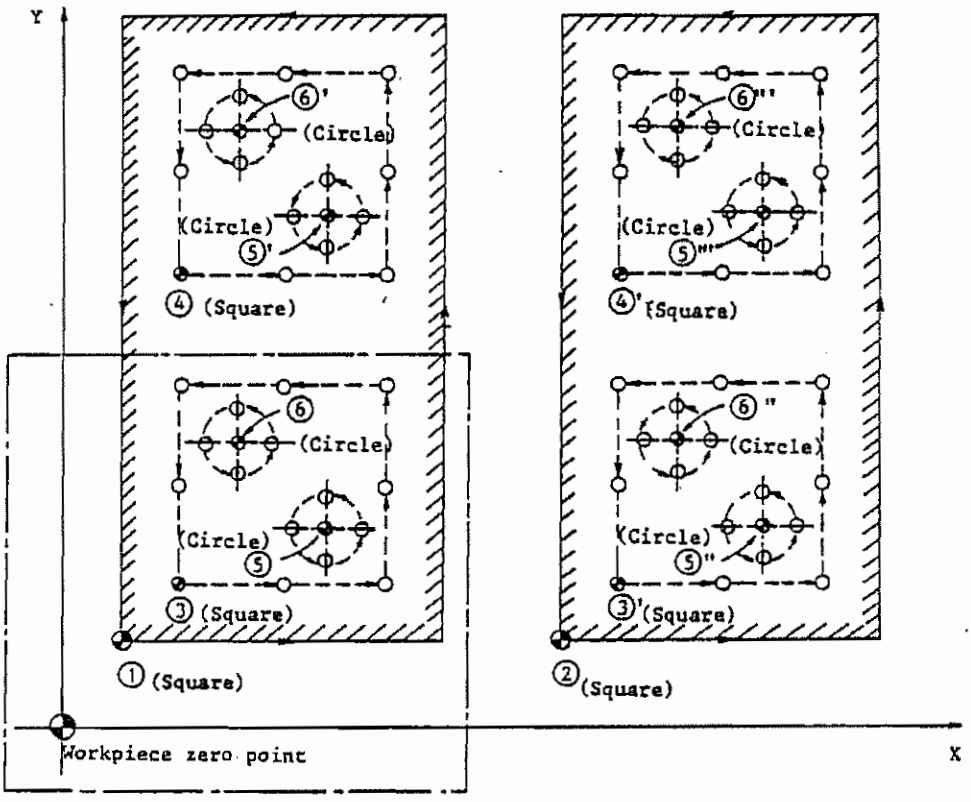
Fig. 1

Commom unit
ofs-1
ofs-2
FRM unit
A Face milling
U 1 Tool 1
Square
Subprogram
D End milling
(plane)
Tool 4
Square
End

Commom unit
ofs-3
ofs-4
B Drilling
U 2 Tool 2
U 3 Tool 3
Square
Subprogram
End

Commom unit
ofs-5
ofs-6
C Drilling
U 2 Tool 2
U 3 Tool 3
Circle
End

Fig.2
Machining pattern

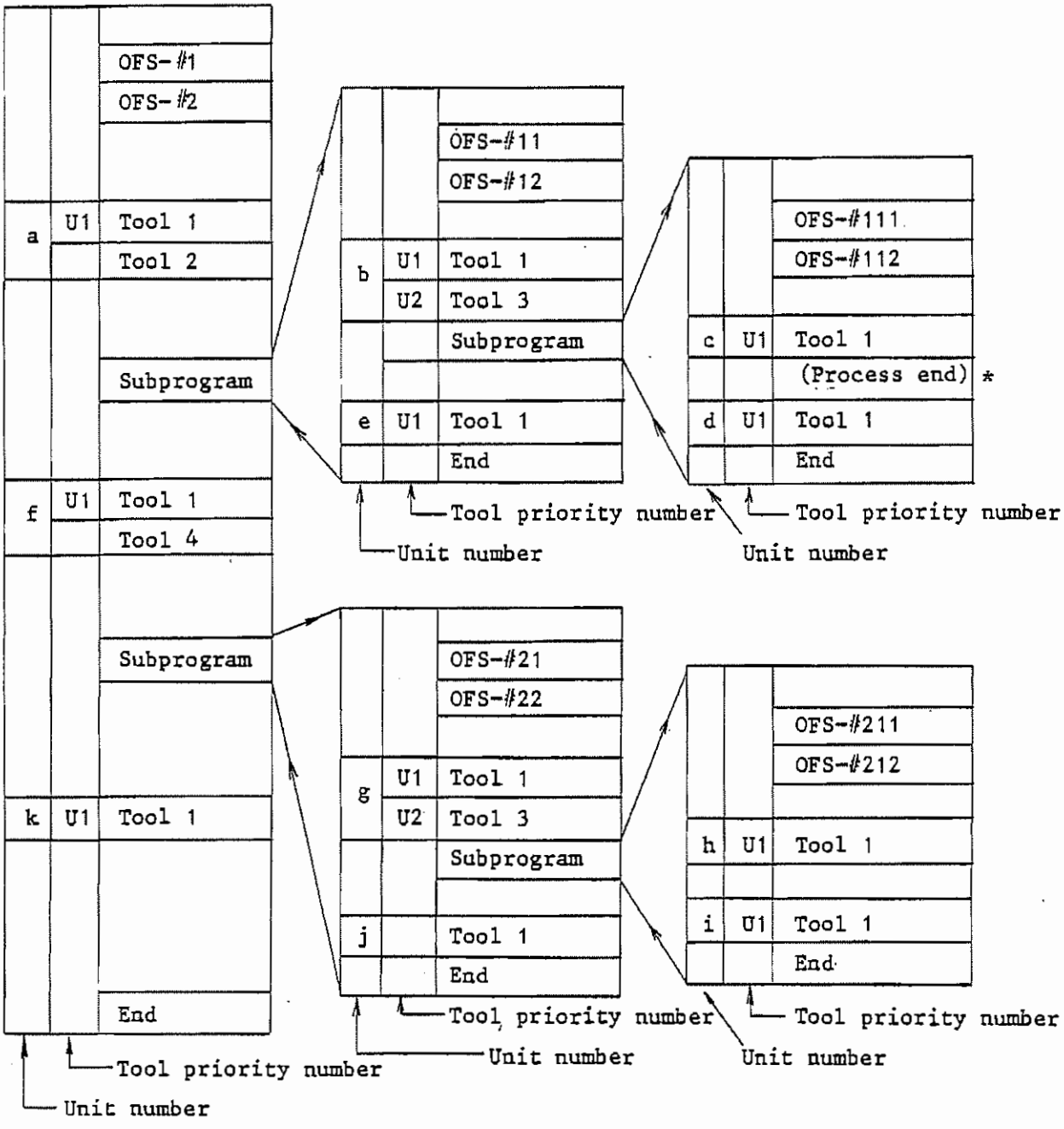


The order of machining by the same tool priority function is as follows:

Order	Offset amount (No. of Fig. 2)		Unit No.	Tool No.
1	ofs-1	①	A	1
2	ofs-2	②	A	1
3	(ofs-1) + (ofs-3)	③	B	2
4	(ofs-1) + (ofs-3) + (ofs-5)	⑤	C	2
5	(ofs-2) + (ofs-3)	⑥	C	2
6	(ofs-1) + (ofs-4)	④	B	2
7	(ofs-1) + (ofs-4) + (ofs-5)	⑤'	C	2
8	(ofs-1) + (ofs-4) + (ofs-6)	⑥'	C	2
9	(ofs-2) + (ofs-3)	③'	B	2
10	(ofs-2) + (ofs-3) + (ofs-5)	⑤''	C	2
11	(ofs-2) + (ofs-3) + (ofs-6)	⑥''	C	2
12	(ofs-2) + (ofs-4)	④'	B	2
13	(ofs-2) + (ofs-4) + (ofs-5)	⑤'''	C	2
14	(ofs-2) + (ofs-4) + (ofs-6)	⑥'''	C	2
15	(ofs-1) + (ofs-3)	③	B	3
16	(ofs-1) + (ofs-3) + (ofs-5)	⑤	C	3
17	(ofs-1) + (ofs-3) + (ofs-6)	⑥	C	3
18	(ofs-1) + (ofs-4)	④	B	3
19	(ofs-1) + (ofs-4) + (ofs-5)	⑤	C	3
20	(ofs-1) + (ofs-4) + (ofs-6)	⑥'	C	3
21	(ofs-2) + (ofs-3)	③'	B	3
22	(ofs-2) + (ofs-3) + (ofs-5)	⑤''	C	3
23	(ofs-2) + (ofs-3) + (ofs-6)	⑥''	C	3
24	(ofs-2) + (ofs-4)	④'	B	3
25	(ofs-2) + (ofs-4) + (ofs-5)	⑤'''	C	3
26	(ofs-2) + (ofs-4) + (ofs-6)	⑥'''	C	3
27	ofs-1	①	D	4
28	ofs-2	②	D	4

Note 1) In the 2*5 multi-piece machining, consider that each machining corresponds to ofs-1, ofs-2, ... in the above example. (The machining in the above example is regarded as multi-piece machining to which offsetting at a equal pitch is applied.)

(ii) More than two subprograms having the same level are involved:



* When the end of tool priority machining number range is entered: (See ②.)

① Order of machining when no end of priority machining number range (by process end, pallet change or drum change) is specified in a subprogram:

	Offset	Unit	Tool
①	#1	a	1
②	#1+#11	b	
③	#1+#11+#111	c	
④	#1+#11+#111	d	
⑤	#1+#11+#112	c	
⑥	#1+#11+#112	d	
⑦	#1+#11	e	
⑧	#1+#12	b	
⑨	#1+#12+#111	c	
⑩	#1+#12+#111	d	
⑪	#1+#12+#112	c	
⑫	#1+#12+#112	d	
⑬	#1+#12	e	
⑭	#1	f	
⑮	#1+#21	g	
⑯	#1+#21+#211	h	
⑰	#1+#21+#211	i	
⑱	#1+#21+#212	h	
⑲	#1+#21+#212	i	
⑳	#1+#22	g	
㉑	#1+#22+#211	h	
㉒	#1+#22+#211	i	
㉓	#1+#22+#212	h	
㉔	#1+#22+#212	i	
㉕	#1	k	
㉖	#2	a	
㉗	#2+#11	b	
㉘	#2+#11+#111	c	
㉙	#2+#11+#111	d	
㉚	#2+#11+#112	c	
㉛	#2+#11+#112	d	
㉜	#2+#12	e	
㉝	#2+#12	b	
㉞	#2+#12+#111	c	1
㉟	#2+#12+#111	d	
㊱	#2+#12+#112	c	
㊲	#2+#12+#112	d	
㊳	#2+#12	e	
㊴	#2	f	
㊵	#2+#21	g	
㊶	#2+#21+#211	h	
㊷	#2+#21+#211	i	
㊸	#2+#21+#212	h	
㊹	#2+#21+#212	i	
㊺	#2+#22	g	
㊻	#2+#22+#211	h	
㊼	#2+#22+#211	i	
㊽	#2+#22+#212	h	
㊾	#2+#22+#212	i	
㊿	#2	k	
(51)	#1+#11	b	3
(52)	#1+#12	b	3
(53)	#1+#21	g	3
(54)	#1+#22	g	3
(55)	#2+#11	b	3
(56)	#2+#12	b	3
(57)	#2+#21	g	3
(58)	#2+#22	g	3
(59)	#1	a	2
(60)	#2	a	2
(61)	#1	f	4
(62)	#2	f	4
(63)	#1+#21	j	1
(64)	#1+#22	j	1
(65)	#2+#21	j	1
(66)	#2+#22	j	1

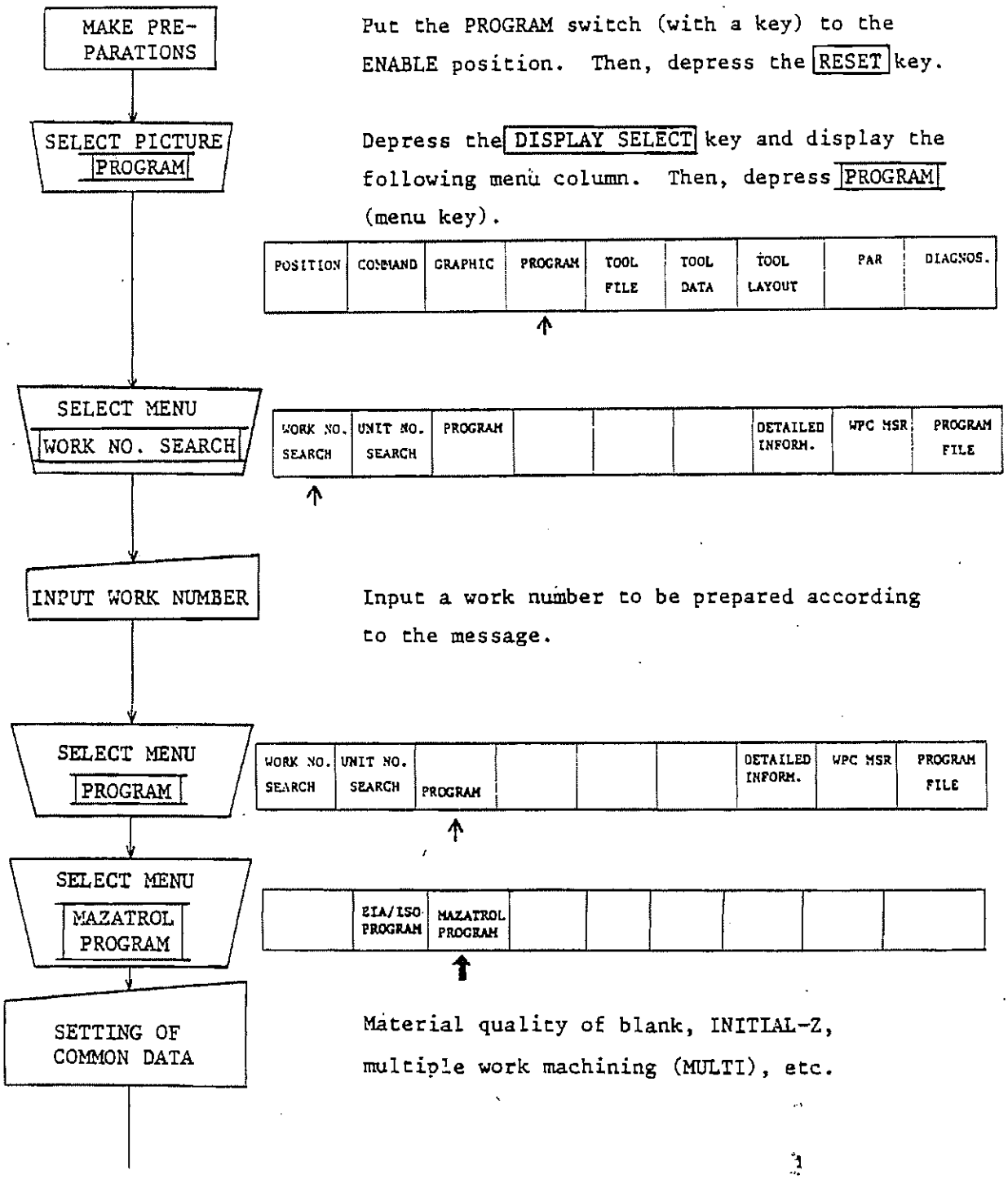
② Order of machining when end of priority machining number range (by process end, pallet change or drum change) is specified in a subprogram:

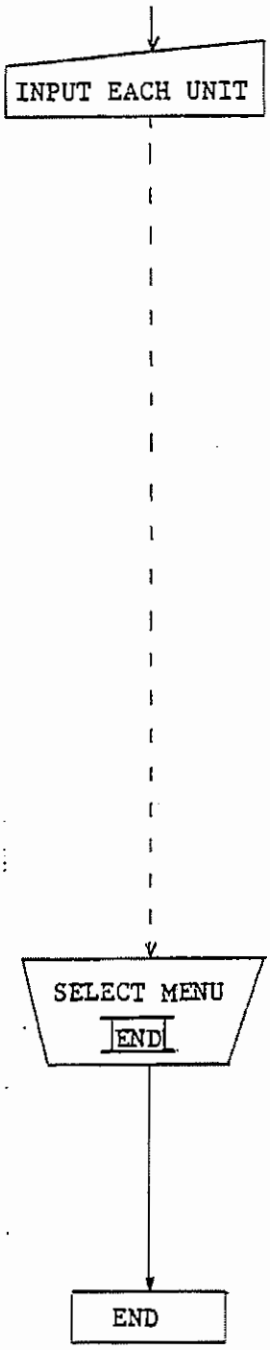
	Offset	Unit	Tool
①	#1	a	1
②	#1+#11	b	
③	#1+#11+#111	c	
④	#1+#11+#112	c	
⑤	#1+#12	b	
⑥	#1+#12+#111	c	
⑦	#1+#12+#112	c	
⑧	#2	a	
⑨	#2+#11	b	
⑩	#2+#11+#111	c	
⑪	#2+#11+#112	c	
⑫	#2+#12	b	
⑬	#2+#12+#111	c	
⑭	#2+#12+#112	c	
⑮	#1+#11	b	3
⑯	#1+#12	b	3
⑰	#2+#11	b	3
⑱	#2+#12	b	3
⑲	#1	a	2
⑳	#2	a	2
Process end ⇨ ㉑	#1+#11+#111	d	1
㉒	#1+#11+#112	d	
㉓	#1+#11	e	
㉔	#1+#12+#111	d	
㉕	#1+#12+#112	d	
㉖	#1+#12	e	
㉗	#1	f	
㉘	#1+#21	g	
㉙	#1+#21+#211	h	
㉚	#1+#21+#211	i	
㉛	#1+#21+#212	h	
㉜	#1+#21+#212	i	
㉝	#1+#22	g	
㉞	#1+#22+#211	h	1
㉟	#1+#22+#211	i	
㊱	#1+#22+#212	h	
㊲	#1+#22+#212	i	
㊳	#1	k	
㊴	#2+#11+#111	d	
㊵	#2+#11+#112	d	
㊶	#2+#11	e	
㊷	#2+#12+#111	d	
㊸	#2+#12+#112	d	
㊹	#2+#12	e	
㊺	#2	f	
㊻	#2+#21	g	
㊼	#2+#21+#211	h	
㊽	#2+#21+#211	i	
㊾	#2+#21+#212	h	
㊿	#2+#21+#212	i	
(51)	#2+#22	g	
(52)	#2+#22+#21	h	
(53)	#2+#22+#21	i	
(54)	#2+#22+#212	h	
(55)	#2+#22+#212	i	
(56)	#2	k	
(57)	#1+#21	g	3
(58)	#1+#22	g	3
(59)	#2+#21	g	3
(60)	#2+#22	g	3
(61)	#1	f	4
(62)	#2	f	4
(63)	#1+#21	j	1
(64)	#1+#22	j	1
(65)	#2+#21	j	1
(66)	#2+#22	j	1

3. PREPARING A PROGRAM

3.1 Procedure

Overview





Basic coordinate unit,
Auxiliary coordinate unit,
Point machining unit,
Line machining unit,
Plane machining unit,
Manual program mode unit
Special unit

Those required for machining must be inputted from these units

While responding to messages,

(1) Input the machining unit and data thereof,
(2) Input the tool development and sequence data, and
(3) Input the machining shape.

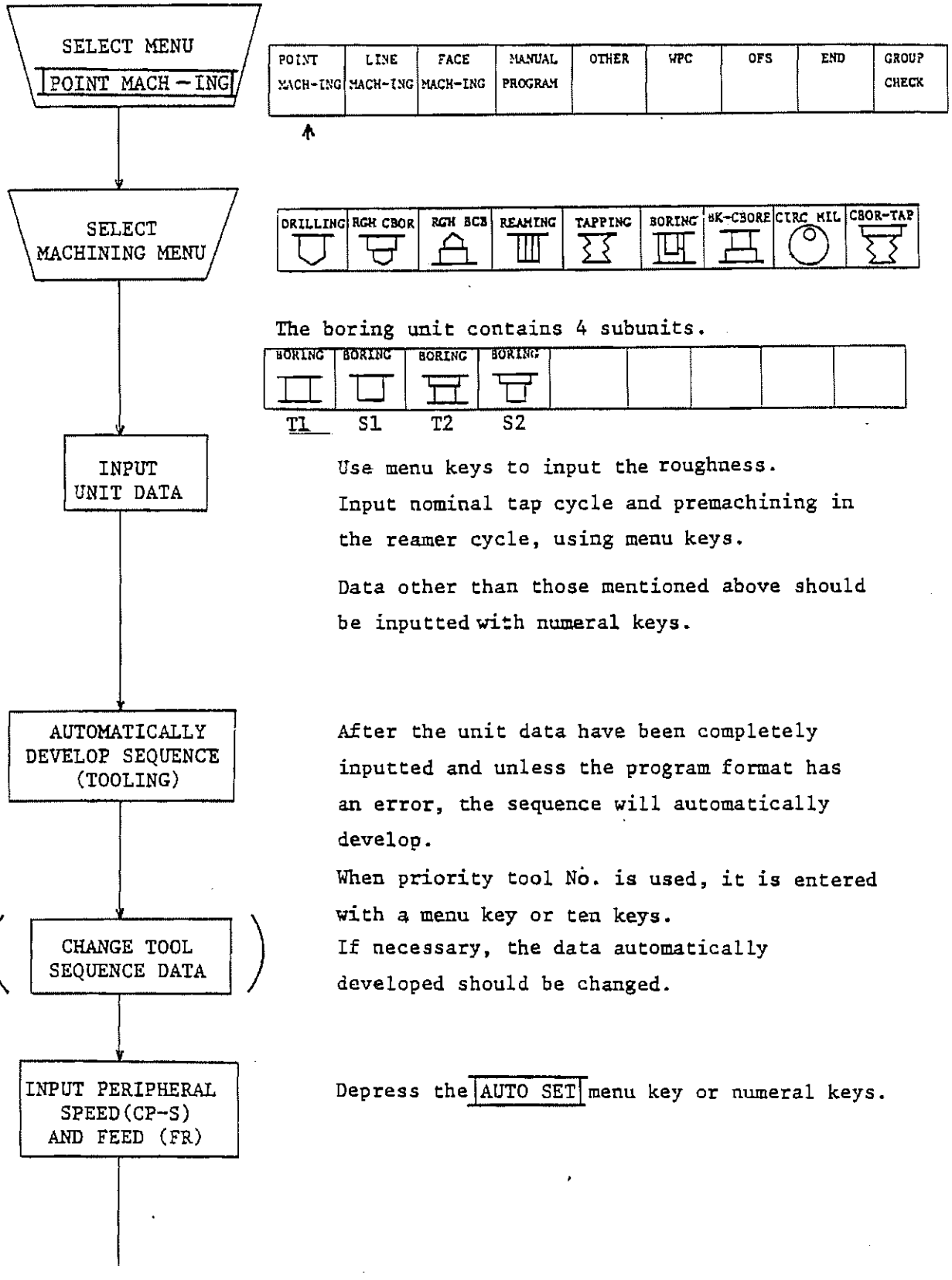
Repeat the above-mentioned operations for input each unit.

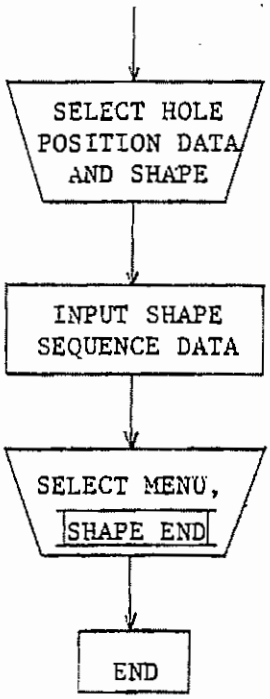
POINT MACH-ING	LINE MACH-ING	FACE MACH-ING	MANUAL PROGRAM	OTHER	WPC	OFS	END	GROUP CHECK

Input the end unit (CONTI. NUMBER).

NOTE . To prepare a program, the work No. must be searched first. It is possible to prepare a new program only when the massage " NEW PROGRAM <PROGRAM> ? " is being displayed on the screen.

(i) Point Machining Program Procedure





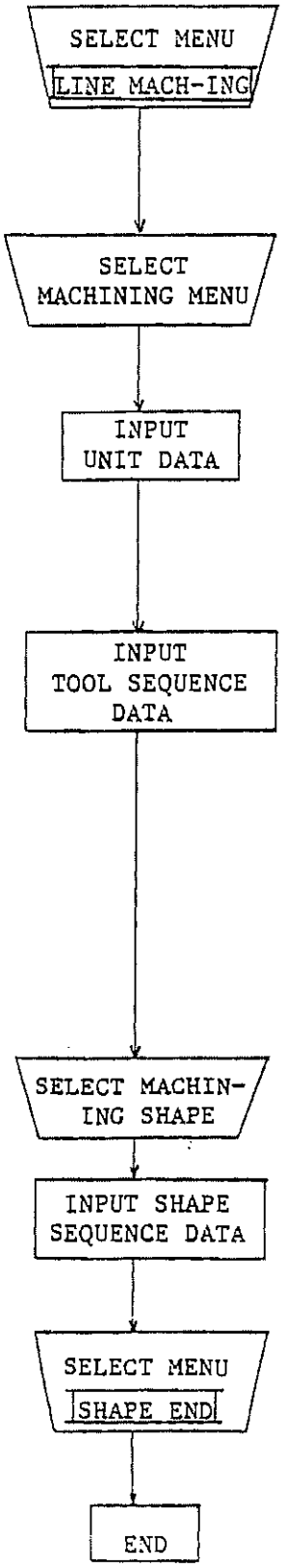
POINT	LINE	SQUARE	GRID	CIRCLE	ARC	CHORD	SHAPE END	CHECK
+								

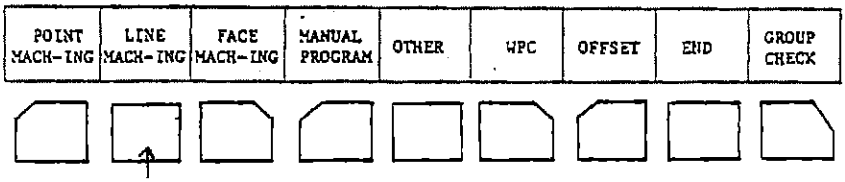
Use the ten keys.

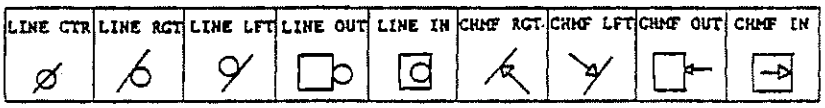
Data on the return point level is entered with ten keys.

POINT	LINE	SQUARE	GRID	CIRCLE	ARC	CHORD	SHAPE END	CHECK
+								

(ii) Line Machining Programming Procedure







Use menu keys to input roughness.
Data other than that should be inputted with the ten keys.

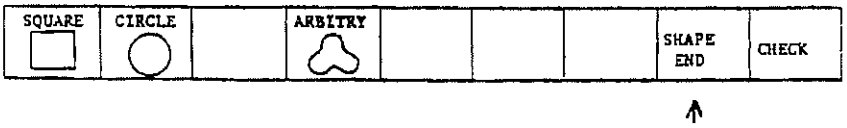
o Once roughness has been inputted, the sequence is dicided into:

Rough cutting only at a roughness of 3 and below, and
Rough cutting and finish (or finish only) at a roughness of 4 and above.

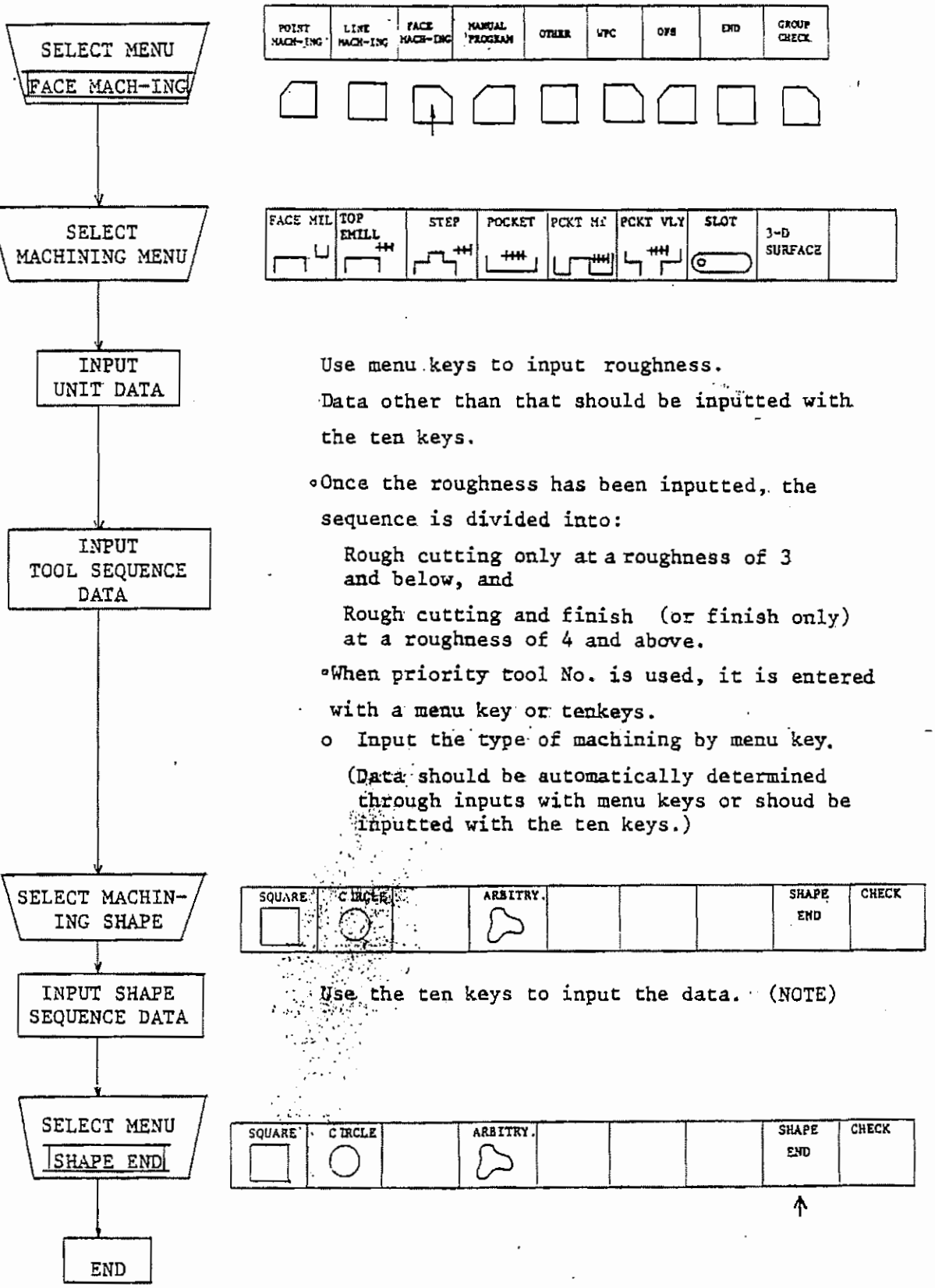
o When priority tool No. is used, it is entered with a menu key or ten keys.

o Use menu keys to input the method.
(Data should be determined automatically through inputs with menu keys or should be inputted with the ten keys.)

Use the ten keys to input the data.



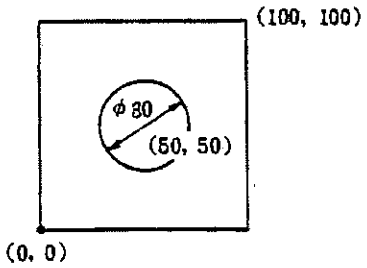
(iii) Face Machining Programming Procedure



NOTE: Whenever two large and small shapes may be defined in a unit to define an end mill valley, pocket peak and pocket valley, always define the larger one first. (Example 1)

If the second or smaller shape is arbitrary, depress menu keys [STARTING POINT] and [LINE] when inputting the shape starting point. Then, check that the number displayed on the picture in blue. (Example 2)

Example 1)



Define as follows:

FIG	PTN	P1X/CX	P1Y/CY	P3X/R	P3Y	CN1	CN2	CN3	CN4
1	SQR	0	0	100	100				
2	CIR	50	50	15	◆	◆	◆	◆	◆

FIG	PTN	P1X/CX	P1Y/CY	P3X/R	P3Y	CN1	CN2	CN3	CN4
1	CIR	50	50	15	◆	◆	◆	◆	◆
2	SQR	0	0	100	100				

If defined as shown above, alarm "382 DEFINED SHAPE TOO SMALL" will appear.

Example 2)

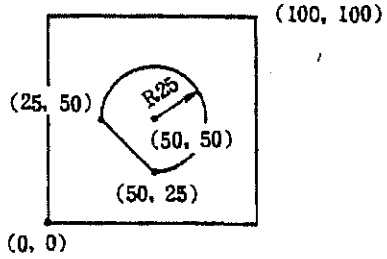


FIG	PTN	P1X/CX	P1Y/CY	P3X/R	P3Y	CN1	CN2	CN3	CN4
1	SQR	0	0	100	100				
	PTN	X	Y	R/θ	I	J	P	CNR	
2	LIN	50	25	← blue					
3	LINE	25	50						
4	CW	50	25	25	50	50	DOWN		

(iv) Manual program mode unit program

PRECAUTIONS

1. Using absolute commands, enter coorkinates for the X-,Y- and Z-axes into the block to which the first axis shift command is given. Otherwise, the tool path will differ from the actual machine operation. (In case where an incremental command is included in the initial position commands of the manual program mode unit.) In principle, the tool moves with the No.1 zero point reckoned as the reference position. Unless any move data have been inputted, moreover, the value of data is reckoned as zero. Besides, if the same tool as the preceeding one has been specified in the program it will move in an incremental stroke from the ending point of the preceeding unit.
2. If two G codes belonging to the same group are entered, only the one which was been entered last will accepted.
3. When the manual program mode unit starts, the modal G codes are automatically put into the following status.(Note 1)

Group A	G00	Fast forward
Group B	G17	XY plane selection
Group C	G40	Tool diameter offset cancel
Group D	G90	Absolute command
Group E	G95	Synchronous feed (/rev)

However, alarm results if the G code specified by the A group (G00, G01, G02 and G03) for the first time is G02 or G03.

4. Deletion of Program Data

To cancel a G, S(speed), or M/B code, move the cursor to the address of the code to be cancelled and enter . CLEAR .

To delete any of the data from 1 through 6, depress the menu key DATA CANCEL after moving the cursor.

G CODE GROUP TABLE

Group A	G00	Fast forward	Modal Ⓐ
	G01	Linear interpolation	Modal A
	G02	Circular interpolation (clockwise)	Modal A
	G03	Circular interpolation (counterclockwise)	Modal A
	G04	Dwell	non-modal
	G28	Return to zero point I	non-modal
	G30	Return to zero point II	non-modal
Group B	G17	Plane selection (XY plane)	Modal Ⓑ
	G18	Plane selection (ZX plane)	Modal B
	G19	Plane selection (YZ plane)	Modal B
Group C	G40	Tool diameter offset cancel	Modal Ⓒ
	G41	Tool diameter offset left (shifted to left when viewed facing forward)	Modal C
	G42	Tool diameter offset right (shifted to right when viewed facing forward)	Modal C
Group D	G90	Absolute command	Modal Ⓓ
	G91	Incremental command	Modal D
Group E	G94	Asynchronous feed command	Modal E
	G95	Synchronous feed command	Modal Ⓔ

Note 1) The modal G code means that which becomes effective until the input of another G code belonging to the same group. The group means gathering of modal G codes interrelated with each other, and the groups are classified into A to E according to their function.

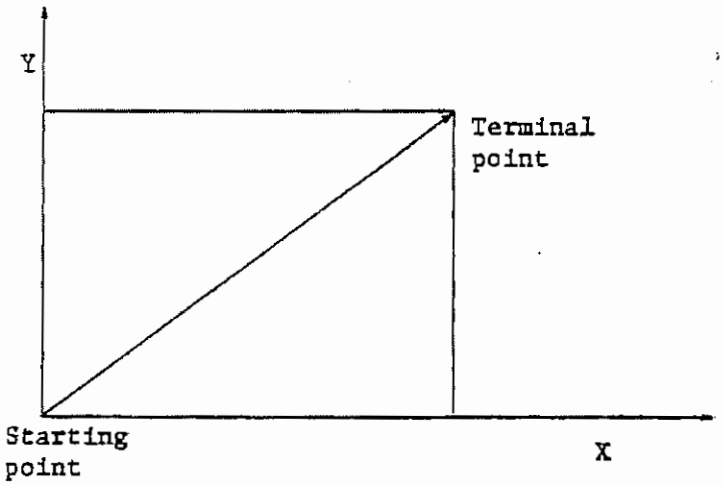
Note 2) A non-modal G code is effective only in the block in which it is entered.

Note 3) Circled G codes are those (modal) automatically selected when initiating manual program mode.
These need not be specified when used at the beginning of a program.

G00

G00 X x_1 Y y_1 Z z_1 A a_1

With the above command, positioning is achieved at quick feed speed. The route of positioning is the shortest linear distance from the starting point to the terminal point.



(1) The number of axes which can simultaneously be driven is determined by the number of axes simultaneously controllable. However, any combination of axes can be adopted to the extent that such simultaneously controllable axes are used.

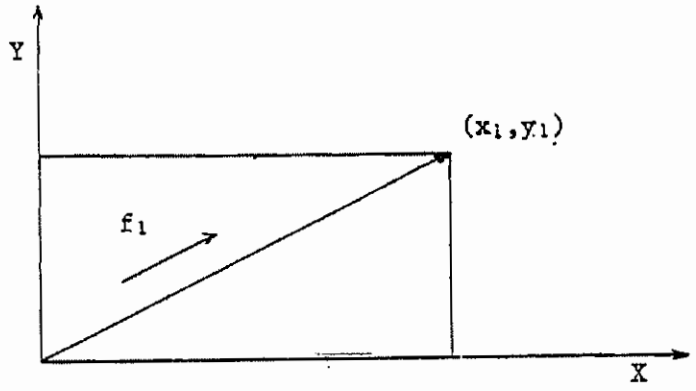
(2) The feed speed is so controlled that the time required is the shortest provided that the feed speed does not exceed the quick feed speed for each axis.

(3) The tool always slows down and stops at the terminal point.

G01

G01 X x_1 Y y_1 Z z_1 A a_1 F f_1

With the above command, linear interpolation is accomplished at the feed speed specified by the F code.

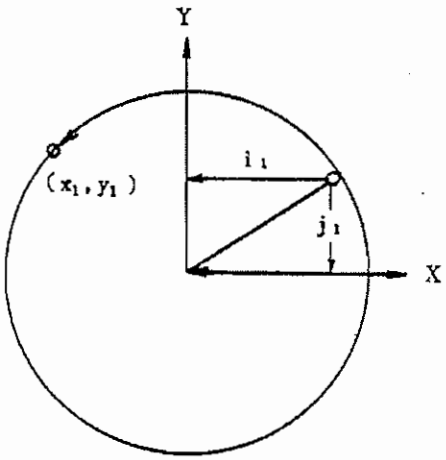


G02, G03

On the plane selected with the plane select G code or addresses of two axes, any circular arc can be selected with the command described below. Multi-quadrant arc interpolation and arc interpolation of additional axis can also be executed.

G02/G03 X x_1 Y y_1 I i_1 J j_1 F f_1

Arc rotation direction
Terminal point coordinates
Feed speed
Incremental stroke from the coordinates of the starting point to the coordinates of the arc center.



G02: Clockwise (CW)
G03: Counterclockwise(CCW)

The plane on which arc interpolation is to be executed is specified with G17, G18 or G19.

G17: Selection of XY plane
G18: Selection of ZX plane
G19: Selection of YZ plane

In addition, arc radius R may be also specified to give the radius in the place of an incremental stroke (I,J,K) up to the center coordinates. The speed in the circumferential direction(rotation direction) is defined by the feed speed at an accuracy of ±1% 1 mm/min.

Note 1. If an I or J command has a contradiction in relations among starting and ending points and center coordinates, alarm 365 (ILLEGAL RADIUS) will result. (Allowable error range: 1mm in the metric system and 0.04 inches in the inch system.)

Note 2. If the cutter diameter compensation mode is changed over with an arc commanded(G40→G41, G40→G42 or G41→G42), an alarm will result.

Helical Cutting

Helical cutting can be effected by entering the linear and pitch commands in the circular cutting program. For details, see "G17, 18 and 19 codes (plane selection)."

G04

The G04 code permits dowel operation.

Setting: After inputting G04 depress the DWELL menu key and input the set value for the dowel (sec.)

Example: Depressing the G04 DWELL 5 INPUT will input G04 D5 in the program.

G17, G18 and G19 Planes selection (XY plane, ZX plane or YZ plane can be selected in that order.)

These codes are used to select the circular cutting arc plane. If any circular arc other than that whose plane has been selected is specified, an alarm will result.

Example:

$$G17, \left\{\begin{matrix}G02\\G03\end{matrix}\right\}, X_1, Y_1, \left\{\begin{matrix}R\_\\I\_, J\_\end{matrix}\right\}$$

$$G18, \left\{\begin{matrix}G02\\G03\end{matrix}\right\}, X_1, Z_1, \left\{\begin{matrix}R\_\\I\_, J\_\end{matrix}\right\}$$

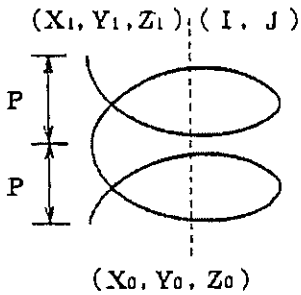
$$G19, \left\{\begin{matrix}G02\\G03\end{matrix}\right\}, Y_1, Z_1, \left\{\begin{matrix}R\_\\J\_, K\_\end{matrix}\right\}$$

Helical Cutting:

Helical cutting is enabled by entering the linear and pitch commands (mm) in the above-mentioned circular cutting program. The circular arc radius R code is invalid only during helical cutting. Prepare the program by selecting the I, J and K central coordinates.

Example:

$$G17, \left\{\begin{matrix}G03\\G02\end{matrix}\right\}, X_1, Y_1, Z_1, I\_, J, P\_$$



(X_1, Y_1, Z_1) : Terminal point coordinares
(I, J) : Incremental stroke to the Center
P : Pitch (mm or inch)

* An error will result if the value of Z_1-Z_0 equals the pitch multipled by an integer.

G28 and G30

When G28 or G30 is given, always enter the coordinate value command in the block concerned(see Note 1).

Note 1:

The coordinate values entered in the block for G28 and G30 constitute relay point coordinates. If the relay point data are not entered, the G28 and G30 commands will not be executed. The axis commanded only returns to the zero point

Example: When G28, Z0 is programmed:

Z0 stands for a relay point only. The Z-axis is, first of all, positioned at the zero point of the workpiece coordinate system in rapid feed. After that, the Z-axis only returns to the zero point.

G40, G41, G42 Tool diameter offset

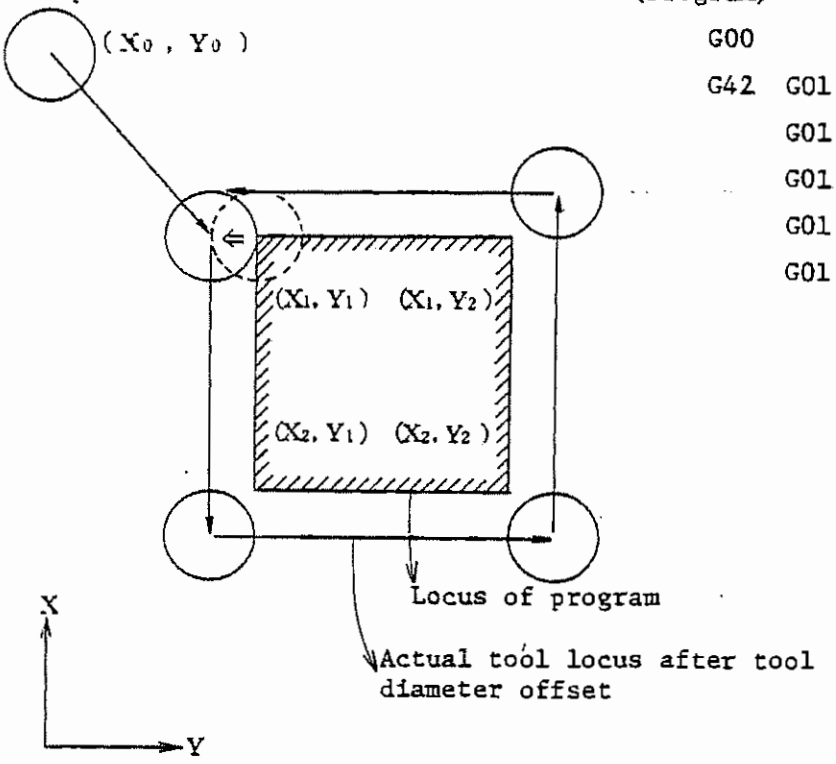
The tool diameter offset is executed in a commanded plane selected. The terminal point coordinate values should be entered after this command has been input.

G41 : Shifts the tool to the left by a distance equal to the tool diameter.

G42 : Shifts the tool to the right by a distance equal to the tool diameter.

G40 : Cancels G41 and G42.

Example:



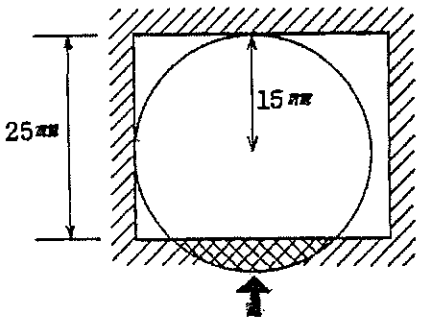
(Program)

```
G00        X0, Y0, Z0
G42  G01   X1, Y1
     G01   X2
     G01        Y2
     G01   X1
     G01        Y1
```

Note 1:

If a workpiece with an inner diameter smaller than the tool's is commanded for cutting the interior of a workpiece, an offset error will result.

Example:



End mill Tool diameter: 30mm

The tool diameter is too great to do the cutting.

⇩

The N/C displays

OFFSET ERROR .

* Use a tool with a smaller diameter.

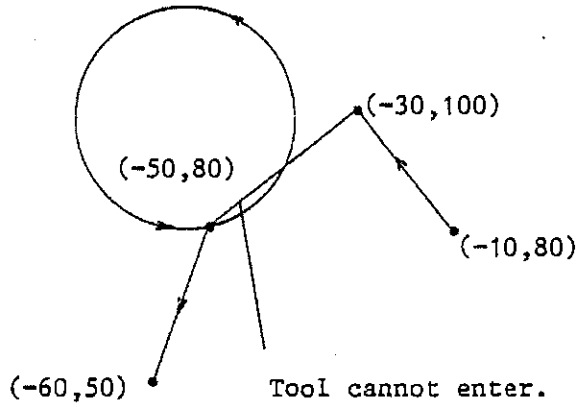
Note 2:

As stated earlier, the terminal point coordinates must be entered after giving G41 and G42. Unless coordinate values are entered in each block without interruption, the tool diameter offset will not be executed properly.
(This is because the N/C will ignore the tool diameter offset in the block when that block does not have a transfer command since the N/C always reads one block in advance.)

Shape Unavailable through Manual Program Cutter Diameter Compensation

(1) Such a shape as illustrated on the right side cannot be moved with the COMPENSATE LEFT command.
The COMPENSATE RIGHT command is applicable.

(This applies to LINE MACHINING LINE LEFT, too.)



UNO	UNIT	TOOL	NOM-Ø	NO.
X X	MANU PRO	E-MILL	20.A	

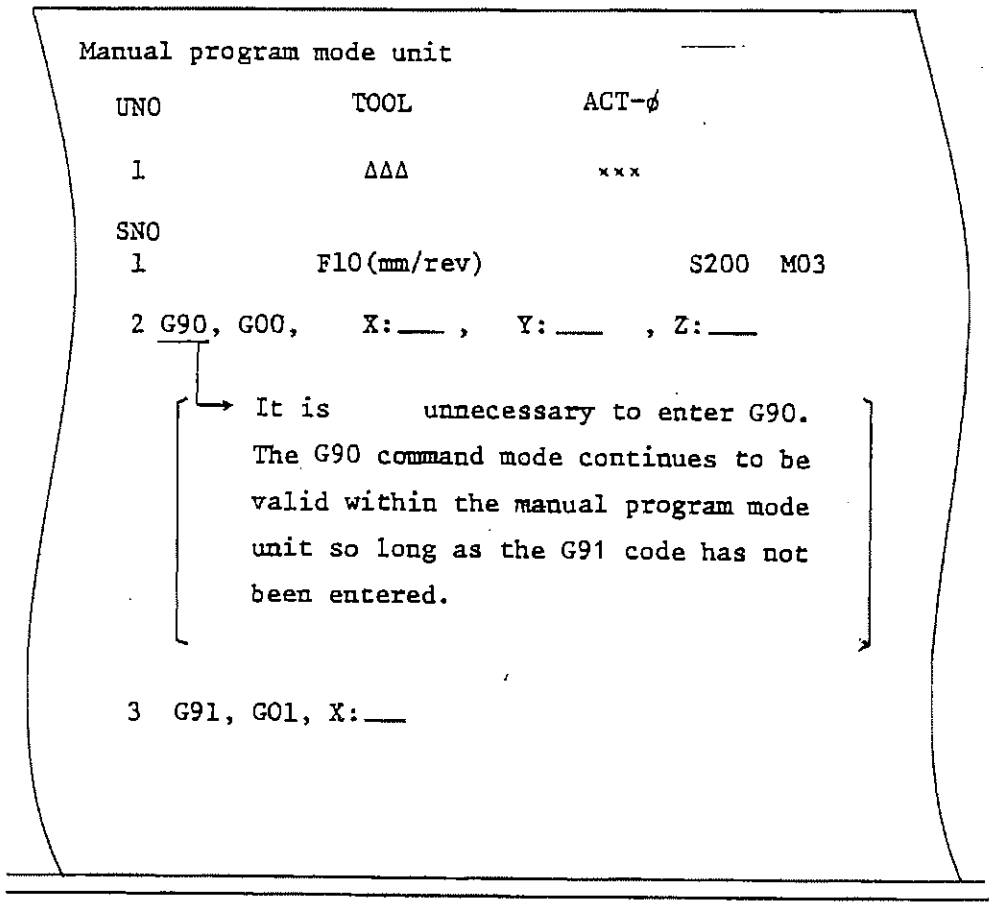
SNO	G1	G2	DATA1	DATA2	DATA3	DATA4 .....
1	0	90	X0	Y0	Z0	
2	1	41	X-10	Y80		
3			X-30	Y100		
4			X-50	Y80		
5	2		X-50		I-10	J20
6	1		X-60	Y50		
⋮						

G90, G91

Never enter the G91 code (incremental code) in the block which has the initial axis shift command for the single action unit. (Instead, enter the absolute code).

In a program which contains more than one manual program mode unit, it is not necessary to execute the G90 command after executing the incremental command in the previous unit because the next unit is automatically given the G90 command (absolute) mode.

Example:



G94, G95

G94: Asynchronous feed(mm/min)

G95: Synchronous feed (mm/rev)

When the power is switched on, the synchronous feed mode is initiated.

When the G94 command is given, the F number is displayed in BLUE on the program screen.

When the G95 code is given, the F number is displayed in YELLOW.

Alarms in Manual Program Mode Unit

(1) No. 315

This alarm is caused, with data insufficient in the arc interpolation under the R command.

(2) No. 331

This alarm is caused, with a specified tool unavailable in the drum. (Tools have not been laid out.)

(3) No. 339

This alarm is caused, with the tool diameter of a specified tool unavailable in tool data (or tool diameter going 0).

(4) No. 363

This alarm is caused unless the block to which the manual program mode unit is commanded to initially move is in the G02 or G03 mode.

(5) No. 365

This alarm is caused in case where there is a contradiction in relations between central coordinates of starting and ending points in the arc interpolation under the (I,J,K) command.

(6) No. 378

This alarm is caused in case where neither (I,J,K) nor R is entered in the arc interpolation, or where the compensation mode is cancelled in the arc interpolation or while the coordinate system is turned on XZ and YZ planes.

(7) No. 379

The alarm is caused in case where identical coordinates value wituout any move entered in the compensation mode (G41 or G42).

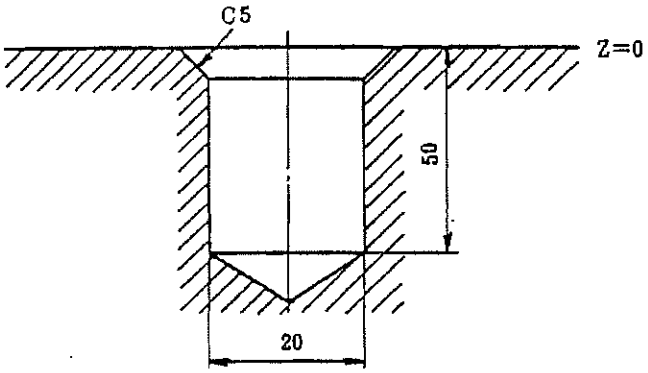
(8) No. 380

This alarm is caused in case where a certain block or blocks has no command in the compensation mode, where neither (I,J,K) nor R is entered under the arc command in the compensation mode,
where three or more blocks of move data are unavailable in the compensation mode,
where the compensation mode is changed over under the arc command in the compensation mode (G41→G42, G42→G41) and/or where the initial block on which the compensation is to act is not in either G00 or G01 mode.

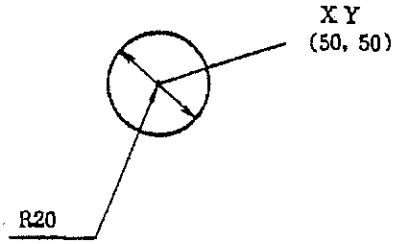
(V) Proctical Examples by Unit

✠ Point Machining (DRILL)

<Sectional Shape>



<Machining Position>



<Program Example>

```
UNO UNIT DIA DEPTH CHMF
XX DRILL 20   50    5                                          (Note1)
SNO  TOOL NOM-ø NO. HOLE-ø HOLE-DEP PRE-DIA PRE-DEP RGH DEPTH C-SP FR   M M
 1  CTR-DR 20        10       •        •        •     •     •   20  0.2
 2  DRILL  20        20      50        •        •   DRILL T50  20  0.201
 3  CHMF   20A       999      0       20       50     •    05  27  0.3
FIG PTN    Z    X    Y    AN1  AN2  T1   T2   F    M    N     P    Q    R
 1  PT     0    50   50    •    •    •    •    •    •    •     0    0    0
                                                          (Note2)(Note3)
                                                                        (note4)
```

Note 1) Depressing menu key HSS AUTO (or CARBIDE AUTO) will automatically determine the data.

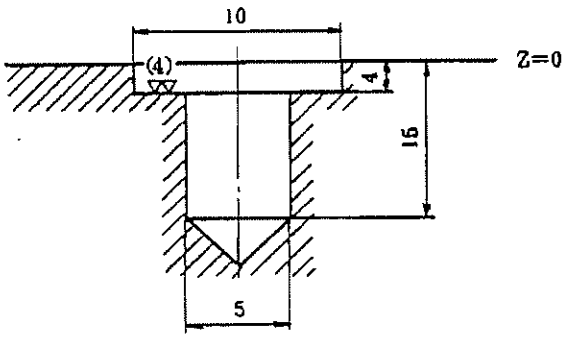
Note 2) When machining a point pattern, P represents a tool path.

Note 3) With Q = 0, a point is machined.
(With Q = 1, positioning only is performed.)

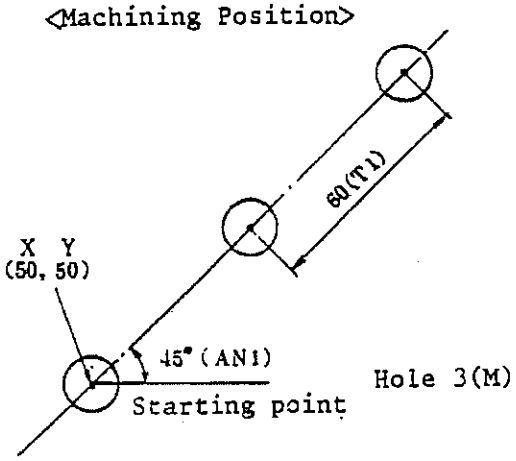
Note 4) Machining is executed after initial point return by R=0.

Point Machining (RGH CBOR)

<Sectional Shape>



<Machining Position>



<Program Example>

UNO	UNIT	CB-DIA	CB-DEP	CHMF	BTM DIA	DEPTH								
XX	RGH	CBOR	10	4	0	4	5	15						
SNO	TOOL	NOM-ϕ	NO.	HOLE-ϕ	HOLE-DEP	PRE-DIA	PRE-DEP	RGH	DEPTH	C-SP	FR	M	M	
1	CTR-DR	20		5	◆	◆	◆	◆	◆	20	0.2			
2	DRILL	5		5	15	◆	◆	DRILL	T15	22	0.066			
3	E-MILL	SA		10	4	5	◆	4	T25	15	0.033			
FIG	PTN	Z	X	Y	AN1	AN2	T1	T2	F	M	N	P	Q	R
1	LIN	0	50	50	45	◆	65	◆	0	3	◆	◆	0	1
									(Note1)				(Note2)	(Note3)

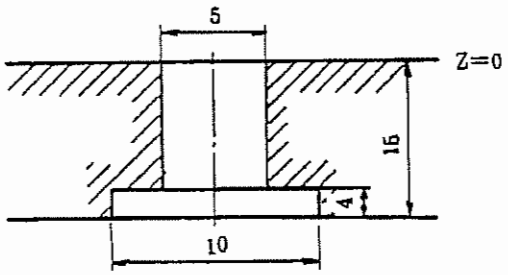
Note 1. With F = 0, holes are spaced at a pitch of 60.

Note 2. With Q = 0, the hole to serve as the starting point is also machined. (With Q = 1, the starting point is positioned only.)

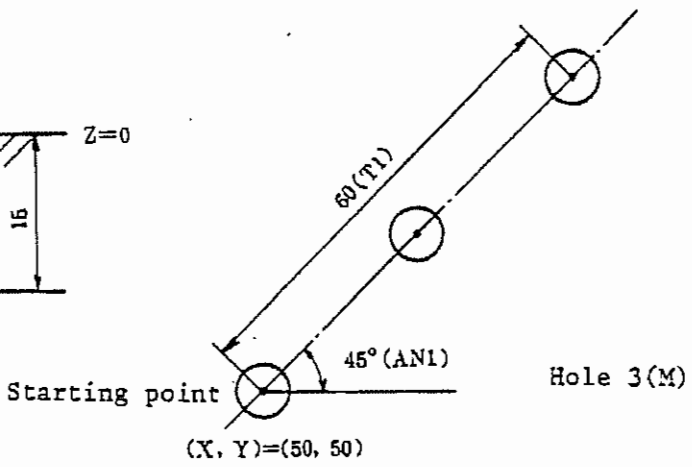
Note 3. Machining is executed after R point return by R=1.

㉘ Point machining(RGH BCB)

<Sectional Shape>



<Machining Position>



<Program Example>

UNO	UNIT	CB-DIA	CB-DEP		DIA	DEPTH	CHMF						
××	RGH	10	4		5	15	0						
	BCB												
SNO	TOOL	NOM-φ	NO.	HOLE-φ	HOLE-DEP	PRE-DIA	PRE-DEP	RGH DEPTH		C-SP	FR	M	M
1	CTR-DR	20		5	◆	◆	◆	◆	◆	20	0.2		
2	DRILL	5		5	16	◆	◆	DRILL	T16	22	0.066		
3	BK FACE	10		10	4	◆	15	◆	◆	19	0.032		

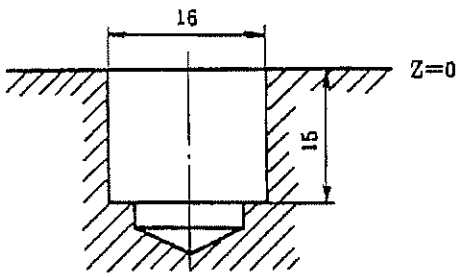
FIG	PTN	Z	X	Y	AN1	AN2	T1	T2	F	M	N	P	Q	R
1	LIN	0	50	50	45	◆	60	◆	1	3	◆	◆	0	0

(Note)

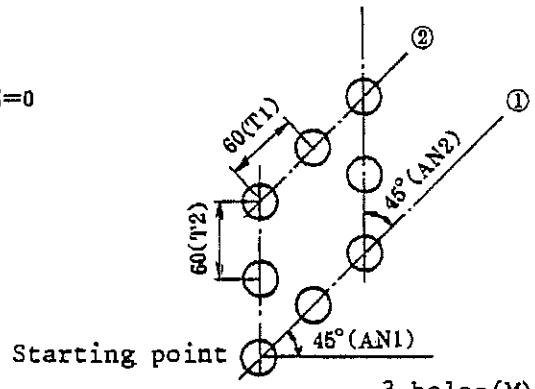
(Note) With F = 1, a row of holes machined has an overall length of 60.

③ Point Machining (REAM)

<Sectional Shape>



<Machining Position>



(X, Y)=(50, 50) 3 holes(M) in direction ①
3 holes(N) in direction ②

<Program Example>

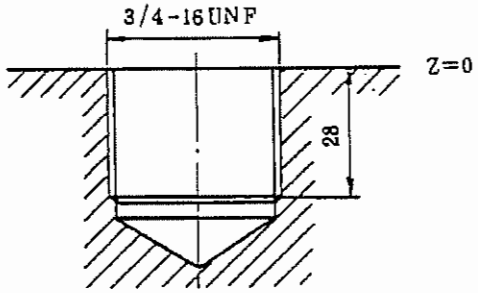
UNO	UNIT	HOLE-φ	HOLE-DEP	CHMF	PRE-REAM	CHP								
X X	REAM	16	15	0.	E-MILL	0								
SNO	TOOL	NOM-φ	NO.	HOLE-φ	HOLE-DEP	PRE-DIA	PRE-DEP	RGH	DEPTH	C-SP	FR	M	M	
1	CTR-DR	20		16	◆	◆	◆	◆	◆	*20	0.2			
2	DRILL	15		15	16	◆	◆	DRILL	T16	25	0.166			
3	E-MILL	10.A		15.8	16	15	◆	4	T0.4	14	0.033			
4	E-MILL	10.B		15.98	16	15.8	◆	4	T0.09	14	0.033			
5	REAM	16		16	15	◆	◆	◆	◆	10	0.576			

FIG	PTN	Z	X	Y	AN1	AN2	T1	T2	F	M	N	P	Q	R
1	SQR	0	50	50	45	45	60	60	0	3	3	0	0	0
												(Note)		

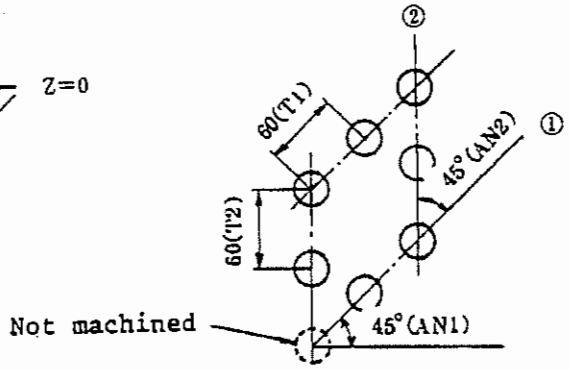
Note) With the machining pattern set to SQR or GRD, P will be:
With P=0, drilling is performed at the three corner points other than the starting one.
With P=1, only positioning is performed at the three corner points other than the starting one.

Point Machining (TAP)

<Sectional Shape>



<Machining Position>



Starting point(X, Y)=(50, 50) 3 holes(M) in direction ①
3 holes(N) in direction ②

<Program Example>

UNO	UNIT	NOM		MAJOR-φ	PITCH	TAP-DEP	CHMF	CHP	*(Note1)				
XX	TAP	3Q-16UN*		19.05	1.587	28	0.6	0					
SNO	TOOL	NOM-φ	NO.	HOLE-φ	HOLE-DEP	PRE-DIA	PRE-DEP	RGH	DEPTH	C-SP	FR	M	M
1	CTR-DR	20		10	◆	◆	◆	◆	◆	20	0.2		
2	DRILL	17.6		17.6	36.935	◆	◆	DRILLT	36.93	25	0.205		
3	TAP	UN3Q-16		19.05	28	◆	◆	◆	P1.587	8	1.587		
FIG	PTN	Z	X	Y	AN1	AN2	T1	T2	F	M	N	P	Q	R
1	SQR	0	50	50	45	45	60	60	0	3	3	0	1	1
														(Note2)

Note 1) How to input a spare unified screw:

(Example) In case of 3/4-16 unified screw:

Depress [Q(1/4) QUARTER] [3] [-] [1] [6] and [INPUT].

(Example 1) In case of 1-1/8 unified screw:

Depress [E(1/8) EIGHTH] [9] [-] [0] [7] and [INPUT].

Do not forget to depress.
([7] only is insufficient.)

How to input a spare tubular screw:

(Example 2)In case of PT 3/8;

Depress [E(1/8) / EIGHTH], [3] and [INPUT].

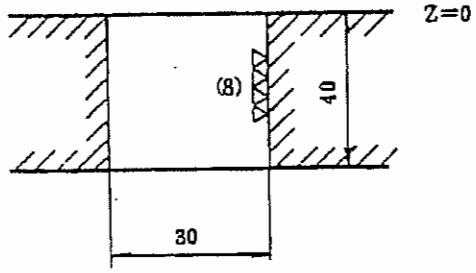
In case of PF 1;

Depress [1] and [INPUT].

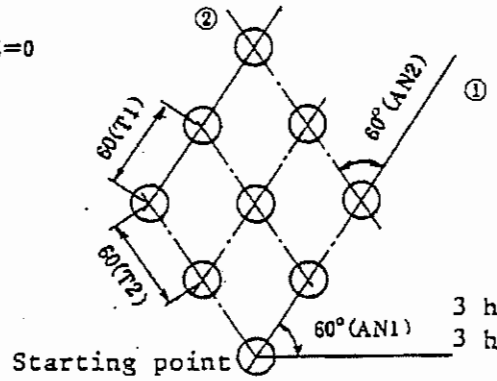
Note 2) With Q = 1, only positioning is performed at the starting point where no machining is to be done.

Point Machining (BORING T1)

<Sectional Shape>



<Machining Position>



3 holes(M) in direction ①
3 holes(N) in direction ②

<Program Example>

UNO	UNIT	DIA	DEPTH	CHMF	WAL	*(Note)							
× ×	BORE T1	30	40	0	8								
SNO	TOOL	NOM-φ	NO.	HOLE-φ	HOLE-DEP	PRE-DIA	PRE-DEP	RGH	DEPTH	C-SP	FR	M	M
1	CTR-DR	20		10	◆	◆	◆	◆	◆	20	0.2		
2	DRILL	24		24	41	◆	◆	DRILL	T41	25	0.228		
3	BORING	28.5		28.5	41	◆	0	0	T225	35	0.127		
4	BORING	29.5		29.5	41	◆	0	0	T0.5	39	0.091		
5	BORING	30		30	41	◆	0	0	T0.25	42	0.066		

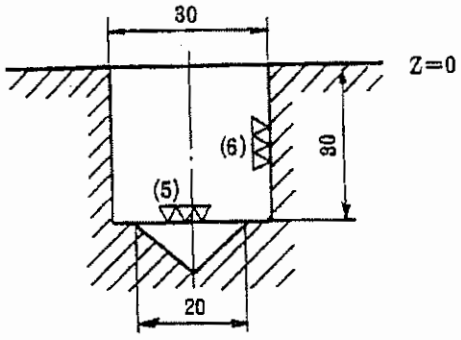
FIG	PTN	Z	X	Y	AN1	AN2	T1	T2	F	M	N	P	Q	R
1	GRD	0	50	50	60	60	60	60	0	3	3	0	0	1

Note 1). For boring, four types of menu are available.
(T1; BORING, T2; BORING , S1; BORING, S2; BORING)

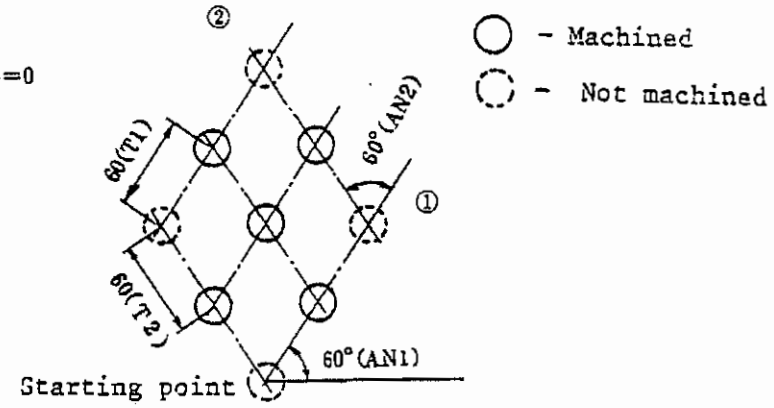
Depress BORING ⇒ BORING .

③ Point Machining (BORING S1)

<Sectional Shape>



<Machining Position>



<Program Example>

3 holes(M) in direction ①
" (N) " ②

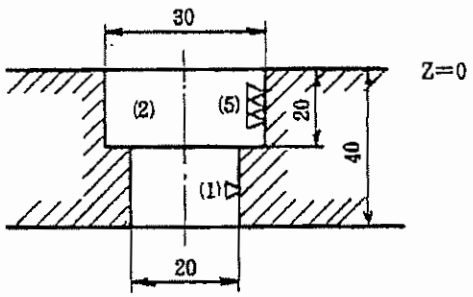
UNO	UNIT	DIA	DEPTH	CHMF	BTM WAL	■	PRE-DIA								
XX	BORE S1	30	30	0	5	6		20							
SNO	TOOL	NOM-φ	NO.	HOLE-φ	HOLE-DEP		PRE-DIA	PRE-DEP	RGH	DEPTH		C-SP	FR	M	M
1	CTR-DR	20		10	◆		◆	◆	◆	◆		20	0.2		
2	DRILL	24		24	28.798		◆	◆	DRILLT	28.79		25	0.228		
3	E-MILL	20.A		24	30		24	◆	5	T0		14	0.104		
4	BORING	28.5		28.5	30		◆	0	5	T2.25		35	0.127		
5	BORING	29.5		29.5	30		◆	0	5	T2.5		39	0.091		
6	BORING	30		30	30		◆	0	5	T0.25		42	0.066		

FIG	PTN	Z	X	Y	AN1	AN2	T1	T2	F	M	N	P	Q	R
1	GRD	0	50	50	60	60	60	60	0	3	3	1 (Note)	1	1

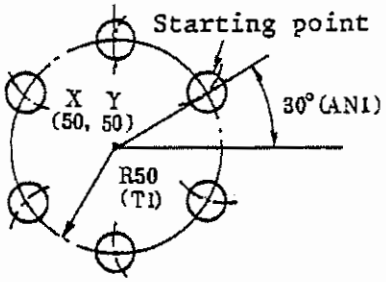
Note) With P = 1 and Q = 1, no machining is performed, but only positioning at the four shape corner points.

■ Point Machining (BORING T2)

<Sectional Shape>



<Machining Position>



Hole 6(M)

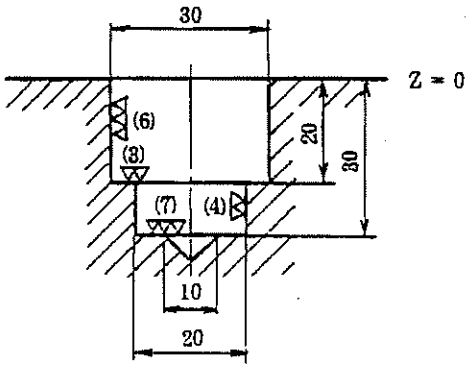
<Program Example>

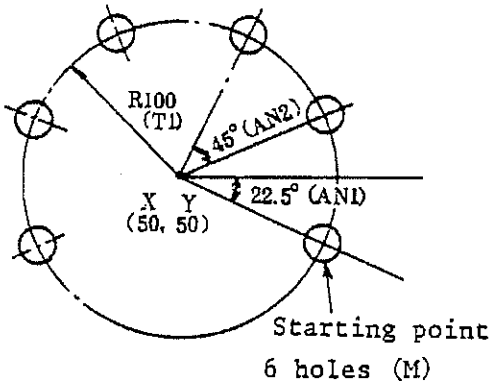
UNO	UNIT	CB-DIA	CB-DEP	CHMF	BTM WAL	DIA	DEPTH	CHMF	WAL				
XX	BORE T2	30	20	0	2	5	20	40	0	1			
SNO	TOOL	NOM-¢	NO. HOLE-¢	HOLE-DEP	PRE-DIA	PRE-DEP	RGH	DEPTH	C-SP	FR	M	M	
1													
1	CTR-DR	20	10	◆	◆	◆	◆	◆	20	0.2			
2	DRILL	14	14	41	◆	◆	DRILL	T41	25	0.16			
3	E-MILL	10.A	24	20	14	◆	2	T5	13	0.033			
4	BORING	20	20	41	◆	20	0	T3	31	0.098			
5	BORING	28.5	28.5	20	◆	◆	2	T2.25	35	0.127			
6	BORING	29.5	29.5	20	◆	◆	2	T0.5	39	0.091			
7	BORING	30	30	20	◆	◆	2	T0.25	42	0.066			

FIG	PTN	Z	X	Y	AN1	AN2	T1	T2	F	M	N	P	Q	R
1	CIR	0	50	50	30	◆	50	◆	◆	6	◆	◆	◆	1

Point Machining (BORING S2)

<Sectional Shape>





<Program Example>

UNC	UNIT	CB-DIA	CB-DEP	CHMF	BTM	WAL	➡	PRE-DIA	DIA DEPTH	CHMF	BTM	WAL	➡
XX	BORE S2	30	20		0	3	6	10	20	30	0	7	4
SNO	TOOL	NOM-φ	NO.	HOLE-φ	HOLE-DEP		PRE-DIA	PRE-DEP	RGH	DEPTH	C-SP	FR	M	M
1	CTR-DR	20		10	◆		◆	◆	◆	◆		20	0.2	
2	DRILL	14		14	28.798		◆	◆	DRILL	T28.79		25	0.16	
3	E-MILL	10.A		14	30		14	◆	7	T0		14	0.033	
4	E-MILL	10.A		24	20		14	◆	3	T5		13	0.033	
5	BORING	19		19	30		◆	20	7	T2.5		30	0.095	
6	BORING	20		20	30		◆	20	7	T0.5		34	0.069	
7	BORING	28.5		28.5	20		◆	0	3	T2.25		35	0.127	
8	BORING	29.5		29.5	20		◆	0	3	T0.5		39	0.091	
9	BORING	30		30	20		◆	0	3	T0.25		42	0.066	

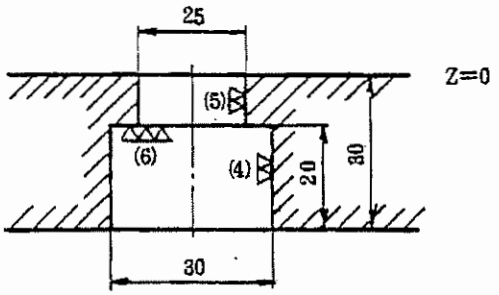
FIG	PTN	Z	X	Y	AN1	AN2	T1	T2	F	M	N	P	Q	R
1	ARC	0	50	50	-22.5	45	100	◆	0	6	◆	◆	0	1

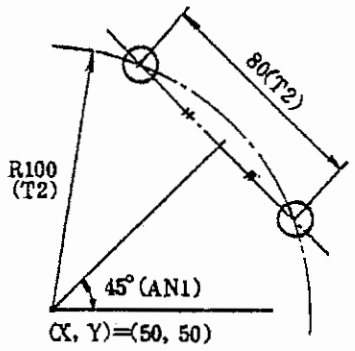
(Note)

Note) With F = 0, an inter-hole arc will have an angle of 45°. (With F = 1, the entire arc will have an angle of 45°.)

Point Machining (BK-CBORE)

<Sectional Shape>





<Program Example>

UNO	UNIT	DIA	DEPTH	BTM	WAL■	PRE-DIA	PRE-DEP	CHMF	WAL■				
XX	BK-CBOR	30	20	6	4	25	30	0	5				
SNO	TOOL	NOM-φ	NO.	HOLE-φ	HOLE-DEP	PRE-DIA	PRE-DEP	RGH	DEPTH	C-SP	FR	M	M
1	CTR-DR	20		10	◆	◆	◆	◆	◆	20	0.2		
2	DRILL	19		19	31	◆	◆	DRILL	T31	25	0.194		
3	BORING	23.5		23.5	31	◆	0	0	T2.25	32	0.11		
4	BORING	24.5		24.5	31	◆	0	0	T0.5	36	0.079		
5	BK-CBOR	29		29	20	24.5	30	6	T2.25	35	0.129		
6	BK-CBOR	30		30	20	24.5	30	6	T0.5	39	0.092		
7	BORING	25		25	11	◆	0	0	T0.25	40	0.058		

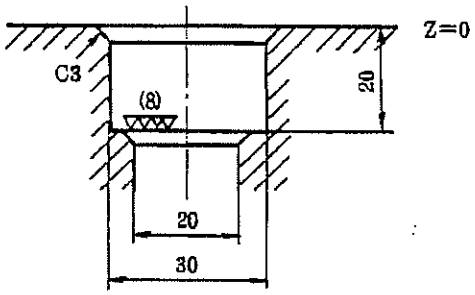
FIG	PTN	Z	X	Y	AN1	AN2	T1	T2	F	M	N	P	Q	R
1	CHD	0	50	50	45	◆	100	80	◆	◆	◆	0	◆	1

(Note)

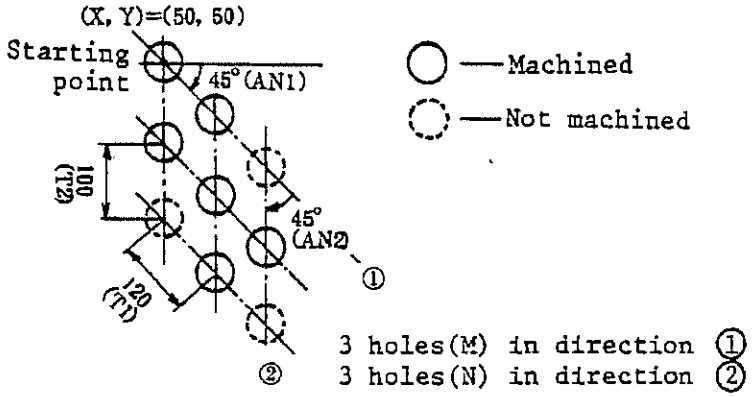
Note) When the machining pattern is set to CHORD, P=0 indicates drilling at points on both sides of the chord; P = 1 on the left side of the chord and P=2 on the right side.
With P = 0, moreover, input the total length of the chord as T1 and with P = 1 or 2, input half the chord as T1.

⑨ Point Machining (CIRC MIL)

<Sectional Shape>



<Machining Position>



<Program Example>

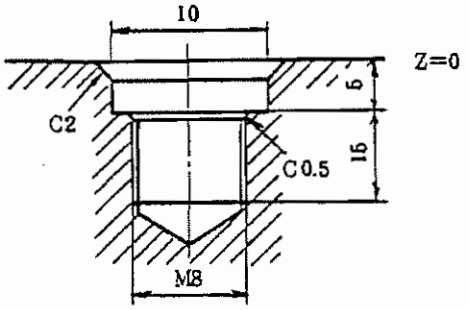
UNO	UNIT	DIA	DEPTH	CHMF	BTM	PRE-DIA	CHMF							
XX	CIRC MIL	30	20	3	8	20	2							
SNO	TOOL NOM-φ	NO.	HOLE-φ	HOLE-DEP	PRE-DIA	PRE-DEP	RGH	DEPTH	C-SP	FR	M	M		
1	E-MILL20.A		30	20	20	◆	8	T5	14	0.12				
2	CHF-M 10.A		999	0	30	20	◆	C3	27	0.3				
3	CHF-M 10.A		30	20	20	999	◆	C3	27	0.3				
FIG	PTN	Z	X	Y	AN1	AN2	T1	T2	F	M	N	P	Q	R
1	GRD	0	50	50	-45	-45	120	100	0	3	3	1	0	1

(Note)

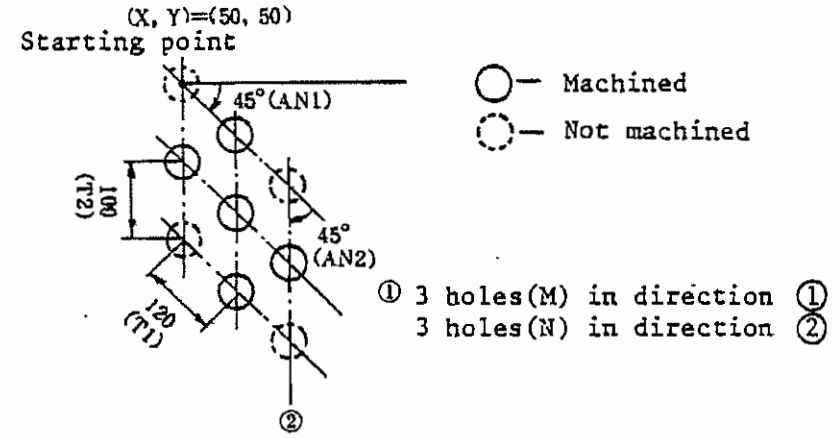
Note) With P = 1 and Q = 0, positioning only is performed at the three corrner points other than the starting one. However, no machining is done.

⊠ Point Machining (CBOR-TAP)

<Sectional Shape>



<Machining Position>



<Program Example>

UNO	UNIT	NOM	MAJOR-φ	PITCH	TAP-DEP	CHMF	CB-DIA	CB-DEP	CHMF	BTM	CHP	
XX	CBOR-TAP	M8	8	1.25	15	0.5	10	5	2	3	1	
SNO	TOOL	NOM-φ	NO.	HOLE-φ	HOLE-DEP	PRE-DIA	PRE-DEP	RGH	DEPTH	C-SP	FR	M M
1	CTR-DR	20		10	◆	◆	◆	◆	◆	20	0.2	
2	DRILL	6.9		6.9	27.5	◆	◆	PCK 1	T3.45	22	0.086	
3	E-MILL	5.A		10	5	6.918	◆	3	T1.54	13	0.024	
4	CHF-M	5.A		999	0	10	5	◆	C2	27	0.3	
5	CHF-M	5.A		10	5	6.918	22.5	◆	C0.5	27	0.3	
6	CHP	VAL 5		◆	◆	◆	◆	◆	◆	◆	◆	
7	TAP	M8		8	20	◆	◆	◆	P1.25	8	1.25	

FIG	PTN	Z	X	Y	AN1	AN2	T1	T2	F	M	N	P	Q	R
1	GRD	0	50	50	-45	-45	120	100	0	3	3	1	1	1

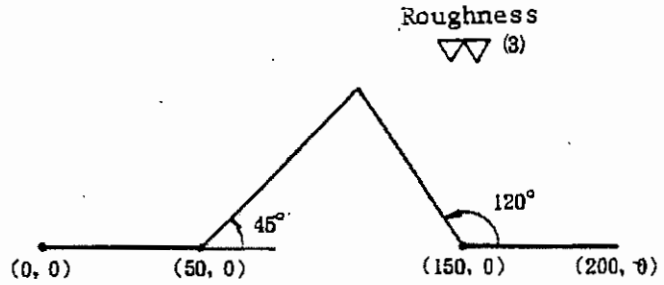
(NOTE)

NOTE : With P=1 and Q=1, four points at corners are positioned only but not machined.

Line Machining (LINE CTR)

(Description)

Line-line machining of an arbitrary shape is performed.



<Program Example>

UNO	UNIT	DEP	SRV-Z	SRV-R	RGH	CHMF	FIN-Z	FIN-R	
XX	LIN CTR	0	5	10	3	◆	0*	◆	*(Note 2)
SNO	TOOL	NOM-ϕ	NO.	APPR-X	APPR-Y	TYPE	ZFD	DEP-Z	WID-R	C-SP	FR	M	M
R1	E-MILL	20.A		?	?	◆	G01	5	◆	17	0.207		
FIG	PTN	X	Y	R/θ	I	J	P	CNR	(Note 3)				
1	LINE	0	0										
2	LINE	50	0										
3	LINE	?	?	45									
4	LINE	150	0	120									
5	LINE	200	0										

Note 1) Define a shape on an arbitrary basis in each of the units (LIN CRT, LIN RGT, LIN LFT, CHMF RGT, CHMF LFT and SLOT).

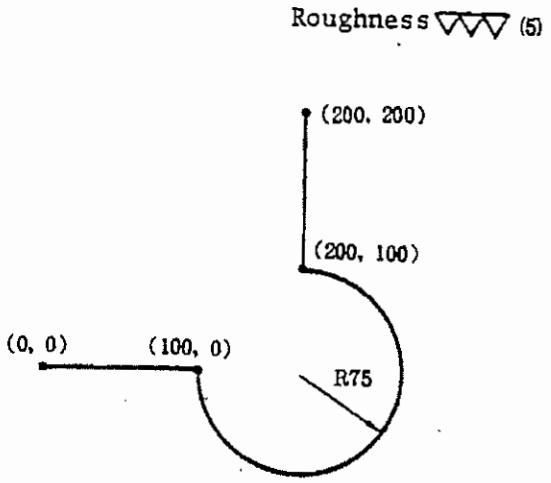
Note 2) Inputting RGH automatically determines FIN-Z.

Note 3) Depressing menu key [HSS AUTO] (or [CARBIDE AUTO]) automatically determines the circumferential speed.

■ Line Machining (LIN RGH)

(Description)

Line-arc machining of an arbitrary shape is performed.



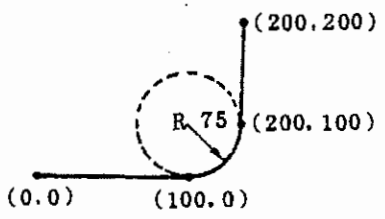
<Program Example>

UNO	UNIT	DEP	SRV-Z	SRV-R	RGH	CHMF	FIN-Z	FIN-R		
XX	LIN RGT	0	5	10	5	◆	0.14	0.14*	*(Note 1)	
SNO	TOOL	NOM-ϕ	NO.	APPR-X	APPR-Y	TYPE	ZFD	DEP-Z	WID-R	C-SP	FR	M	M
R1	E-MILL	20.A		?	?	◆	G01	4.86	◆	17	0.207		
F2	E-MILL	20.A		?	?	◆	G01		◆	17	0.32		
FIG	PTN	X	Y	R/θ	I	J	P	CNR					
1	LINE	0	0										
2	LINE	100	0	(Note 2)									
3	CCW	200	100	-75									
4	LINE	200	200										

Note 1) Inputting RGH automqtically determines FIN-Z and FIN-R.

Note 2) Either CW or CCW, an arc with angle of 180° and above requires an input of a negative value(-) as radius R.
However, the arc center values I and J may be positive (+)

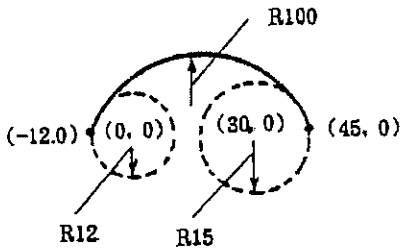
With 75 (positive) inputted as R, meanwhile, the shape will be as illustrated on the right.



- Line Machining (LIN LFT)

Two arcs, without crossing are connected with a corner radius given. This description relates to a program where R > 0. See the note below in which R < 0 is explained.

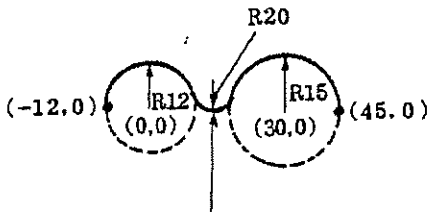
CW arc will result, with R>0



<Program Example>

UNO	UNIT	DEP	SRV-Z	SRV-R	RGH	CHMF	FIN-Z	FIN-R				
XX	LIN LFT	0	5	10	3	◆	0	0				
SNO	TOOL NOM-ϕ	NO.	APPR-X	APPR-Y	TYPE	ZFD	DEP-Z	WID-R	C-SP	FR	M	M
R1	E-MILL 20.A		?	?	◆	G01	5	◆	17	0.207		
FIG	PTN	X	Y	R/θ	I	J	P	CNR				
1	LINE	-12	0									
2	CW	?	?	12	0	0		R100				
3	CCW	45	0	15	30	0		(Note)				

Note) With R< 0, the shape will be as illustrated below. In the below example, R is -20.



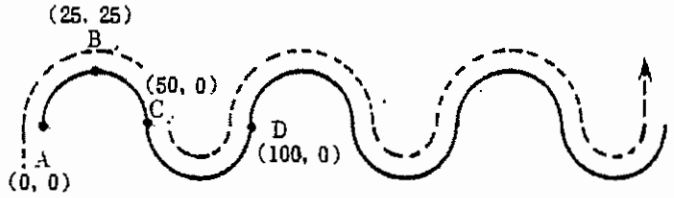
CCW arc will result, with R < 0.

⊕ Line Machining (LIN LFT)

Roughness ▽▽ (3)

Graphic Shift Function

The graph shown on the right can be defined by setting points A, B, C and D between SH and REP in the program shape sequence.



(- - - Tool path)

Arc-arc machining is performed in an arbitrary shape.

<Program Example>

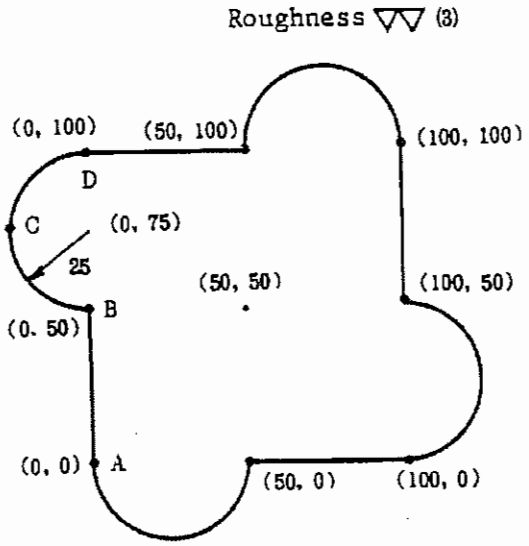
UNO	UNIT	DEP	SRV-Z	SRV-R	RGH	CHMF	FIN-Z	FIN-R				
XX	LIN LFT	0	5	10	3	◆	0	0				
SNO	TOOL	NOM-ϕ	NO.	APPR-X	APPR-Y	TYPE	ZFD	DEP-Z	WID-R	C-SP	FR	M M
R1	E-MILL	20.A		?	?	◆	G01	5	◆	17	0.207	

FIG	PTN	X	Y	R/θ	I	J	P	CNR
1	FIG-SH	◆	◆	◆	◆	◆	3	◆
2	LINE	0	0					
3	CW	25	25	25				
4	CW	50	0	25				
5	CCW	100	0	25				
	PEP-EN	◆	◆	◆	◆	◆	◆	◆

Line Machining (LIN OUT)

Graphic Rotating Function

The graph illustrated on the right can be defined by setting points A, B, C and D between CW and REP in the program shape sequence.



<Program Example>

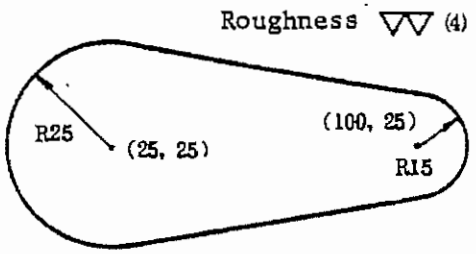
UNO	UNIT	DEP	SRV-Z	SRV-R	RGH	CHMF	FIN-Z	FIN-R		
XX	LIN OUT	0	5	10	3	◆	0	0		
SNO	TOOL	NOM-ϕ	NO.	APPR-X	APPR-Y	TYPE	ZFD	DEP-Z	WID-R	C-SP	FR
R1	E-MILL	20.A		?	?	CW	G01	5	◆	17	0.207
FIG	PTN	X	Y	R/θ	I	J	P	CNR			
1	CW-SH	◆	◆	?	50	50	4	◆			
2	LINE	0	0								
3	LINE	0	50								
4*	CW	0	100	25	0	75	*(NOTE)				
5	PEP-EN	◆	◆	◆	◆	◆	◆	◆			

Note) For CW and CCW arcs within the graphic rotating function, other than a 90° one input all of X, Y, R/θ, I and J.

Line Machining (LIN IN)

(Description)

An arbitrary shape is programmed so that an arc meets a line.



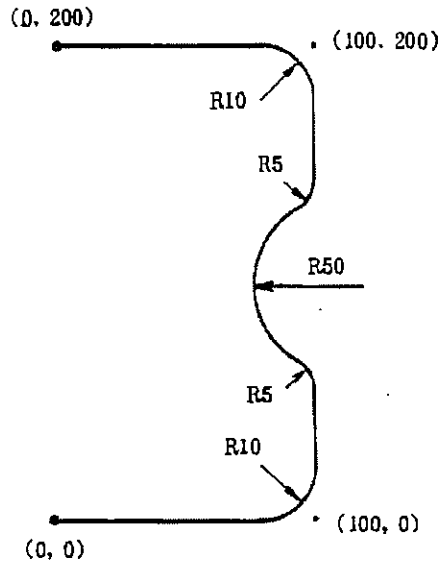
<Program Example>

UNO	UNIT	DEP	SRV-Z	SRV-R	RGH	CHMF	FIN-Z	FIN-R				
XX	LIN IN	5	5	10	4	◆	0.6	0.6				
SNO	TOOL	NOM-ϕ	NO.	APPR-X	APPR-Y	TYPE	ZFD	DEP-Z	WID-R	C-SP	FR	M M
R1	E-MILL	20.A		?	?	CW	G01	4.4	◆	19	0.296	
F2	E-MILL	20.A		?	?	CW	G01	◆	◆	19	0.296	
FIG	PTN	X	Y	R/θ	I	J	P	CNR				
1	CW	?	?	25	25	25						
2	LINE	?	?									
3	CW	?	?	15	100	25						
4	LINE	?	?									

Line Machining (CHMF RGT)

(Description)

An arbitrary shape is machined with a turning radius at the corners.



<Program Example>

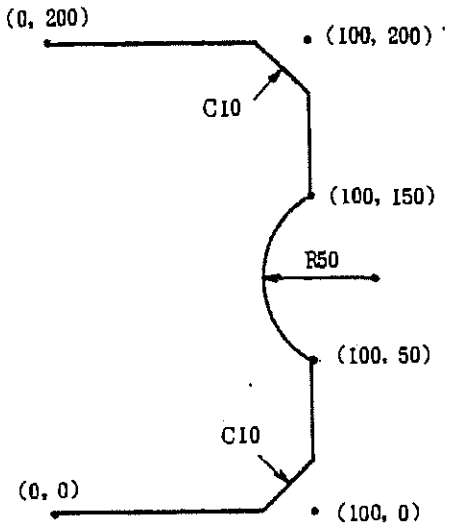
UNO	UNIT	DEP	INTER-Z	INTER-R	RGH	CHMF					
XX	CHMF RGH	0	99	99	◆	5					
SNO	TOOL NOM-ϕ	NO.	APPR-X	APPR-Y	TYPE	ZFD	DEP-Z	WID-R	C-SP	FR	M M
1	CHM-F	20.A	?	?	◆	G01	◆	◆	27	0.3	
FIG	PTN	X	Y	R/θ	I	J	P	CNR			
1	LINE	0	200								
2	LINE	100	200					R10*(NOTE)			
3	LINE	100	150					R5			
4	CCW	100	50	50				R5			
5	LINE	100	0					R10			
6	LINE	0	0								

Note) If CNR (corner) is to be inputted as a radius, input numerals only. If CNR as C is to be inputted, depress menu key [CORNER C] and then input numerals. (See the next page.)

Line Machining (CHMF LFT)

(Description)

An arbitrary shape is machined with C at the corners.



<Program Example>

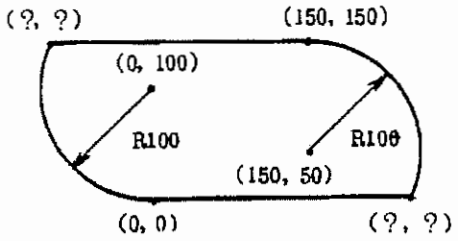
UNO	UNIT	DEP	INTER-Z	INTER-R	RGH	CHMF						
XX	CHMF LFT	0	99	99	◆	5						
SNO	TOOL	NOM-φ	NO.	APPR-X	APPR-Y	TYPE	ZFD	DEP-Z	WID-R	C-SP	FR	M M
1	CHM-F	20.A		?	?	◆	G01	◆	◆	27	0.3	
FIG	PTN	X	Y	R/θ	I	J	P	CNR				
1	LINE	0	0									
2	LINE	100	0					C10*(Note)				
3	LINE	100	50									
4	CW	100	150	50								
5	LINE	100	200					C10				
6	LINE	0	200									

Note) Depress [CORNER C], [1], [0] and [INPUT].

Line Machining (CHMF OUT)

(Description)

An arbitrary shape is programmed with an unknown crossing with a subsequent graph.



<Program Example>

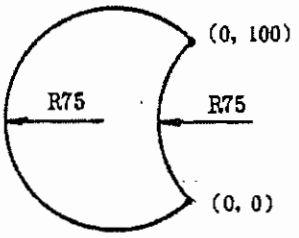
UNO	UNIT	DEP	INTER-Z	INTER-R	RGH	CHMF						
XX	CHMF OUT	0	99	99	◆	7						
SNO	TOOL	NOM-ϕ	NO.	APPR-X	APPR-Y	TYPE	ZFD	DEP-Z	WID-R	C-SP	FR	M M
1	CHM-F	20.A		?	?	CCW	G01	◆	◆	27	0.3	
FIG	PTN	X	Y	R/θ	I	J	P	CNR				
1	LINE	0	0									
2	LINE	?	?	0			RGT*(Note)					
3	CCW	150	150	100	150	50	RGT					
4	LINE	?	?	0			LFT					
5	CCW	0	0	100	0	100						

Note) If there are two crossings with a graph subsequently defined (LINE, CW and CCW), use a menu key (UP, DOWN, RIGHT and LEFT) to specify either of the crossings.

Line Machining (CHMF IN)

(Description)

In an arbitrary shape, the arc between two points is defined.



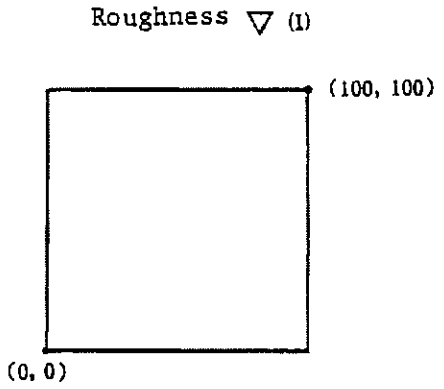
<Program Example>

UNO	UNIT	DEP	INTER-Z	INTER-R	RGH	CHMF						
XX	CHMF IN	0	99	99	◆	7						
SNO	TOOL	NOM-φ	NO.	APPR-X	APPR-Y	TYPE	ZFD	DEP-Z	WID-R	C-SP	FR	M M
1	CHMF	20.A		?	?	CW	G01	◆	◆	27	0.3	
FIG	PTN	X	Y	R/θ	I	J	P	CNR				
1	CW	0	100	75								
2	CCW	0	0	-75								

Face Machining (F-MILL)

(Description)

A fixed square shape is programmed.



<Program Example>

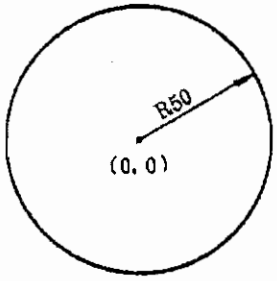
UNO	UNIT	DEP	SRV-Z	SRV-R	BTM	WAL	FIN-Z	FIN-R					
XX	F-MILL	0	5	◆	1	◆	0	◆					
SNO	TOOL	NOM-ϕ	NO.	APPR-X	APPR-Y	TYPE	ZFD	DEP-Z	WID-R	C-SP	FR	M	M
R1	F-MILL	50.A		?	?	BI-DIA-X	G01	5	35	24	1.35		
FIG	PTN	P1X/CX	P1Y/CY	P3X/R	P3Y	CN1	CN2	CN3	CN4				
1	SQR	0	0	100	100								

Face Machining (TOP EMIL)

(Description)

A fixed circular shape is programmed.

Roughness ▽▽▽ (9)



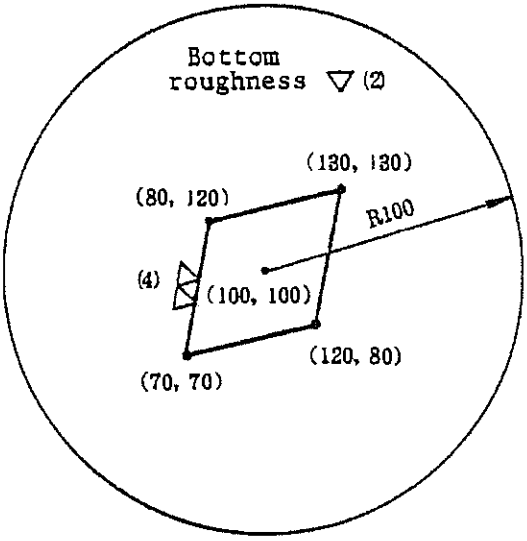
<Program Example>

UNO	UNIT	DEP	SRV-Z	SRV-R	BTM	WAL	FIN-Z	FIN-R					
XX	TOP EMIL	0	5	◆	9	◆	0.101	◆					
SNO	TOOL	NOM-ϕ	NO.	APPR-X	APPR-Y	TYPE	ZFD	DEP-Z	WID-R	C-SP	FR	M	M
R1	E-MILL	20.A		?	?	CW	G01	10	12	12	0.094		
F2	E-MILL	20.A		?	?	CW	G01	◆	12	14	0.0448		
FIG	PTN	P1X/CX	P1Y/CY	P3X/R	P3Y	CN1	CN2	CN3	CN4				
1	CIR	0	0	50	◆	◆	◆	◆	◆				

⊞ Face Machining (STEP)

(Description)

How to define an external and an internal shape. In the shape sequence, the external shape should be inputted first. Define external and internal shapes as fixed and arbitrary, respectively.



<Program Example>

UNO	UNIT	DEP	SRV-Z	SRV-R	BTM	WAL	FIN-Z	FIN-R				
XX	STEP	0	10	◆	2	4	0	0.6				
SNO	TOOL	NOM-ϕ	NO.	APPR-X	APPR-Y	TYPE	ZFD	DEP-Z	WID-R	C-SP	FR	M M
R1	E-MILL	20.A		?	?	CW	G01	10	12	14	0.135	
F2	E-MILL	20.A		?	?	CW	G01	◆	12	16	0.448	
FIG	PTN	P1X/CX	P1Y/CY	P3XR	P3Y	CN1	CN2	CN3	CN4			
1	CIR	100	100	100	◆	◆	◆	◆	◆			
FIG	PTN	X	Y	R/θ	I	J	P	CNR				
2	LINE	70	70									
3	LINE	80	120									
4	LINE	130	130									
5	LINE	120	80									

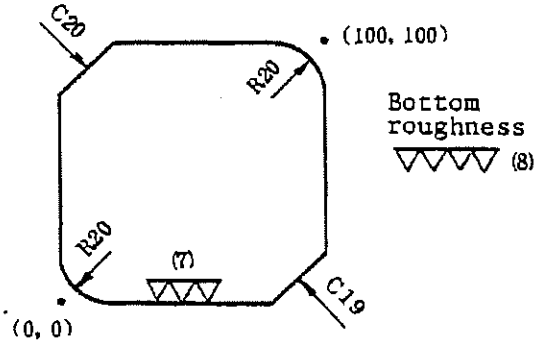
Note) If the second shape (internal shape) is arbitrary, depress menu keys [STARTING POINT] and [LINE] in that order when inputting the starting point.

Values displayed in blue indicates the starting point.

Face Machining (POCKET)

(Description)

The graph in which a fixed shape contains both a corner radius and corner C is to be programmed.



<Program Example>

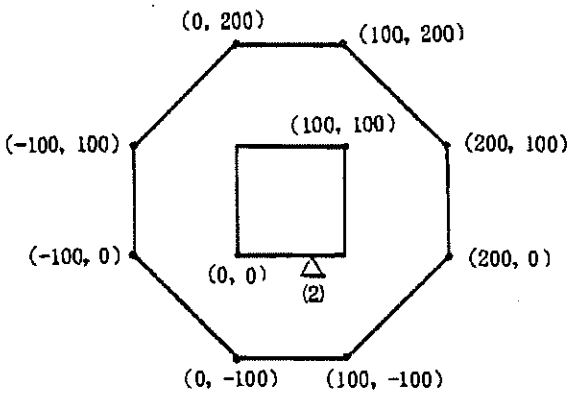
UNO	UNIT	DEP	SRV-Z	SRV-R	BTM	WAL	FIN-Z	FIN-R				
XX	POCKET	0	5	◆	8	7	0.144	0.206				
SNO	TOOL	NOM-φ	NO.	APPR-X	APPR-Y	TYPE	ZFD	DEP-Z	WID-R	C-SP	FR	M M
R1	E-MILL	20.A		?	?	CCW	G01	4.856	12	14	0.135	
F2	E-MILL	20.A		?	?	CCW	G01	◆	12	14	0.163	
FIG	PTN	P1X/CX	P1Y/CY	P3X/R	P3Y	CN1	CN2	CN3	CN4			
1	SQR	0	0	100	100	R20	C20	R20	C19			

Face Machining (POKT MT)

(Description)

Define external and internal shapes as arbitrary and fixed, respectively. In the shape sequence, define the external shape first.

The external shape is defined using the graphic rotating function.



Roughness ▽▽ (3)

<Program Example>

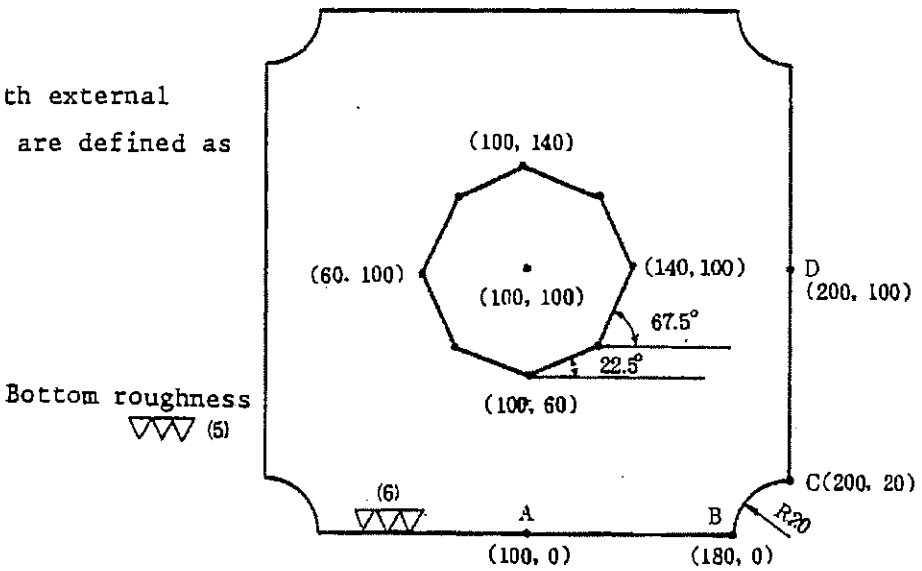
UNO	UNIT	DEP	SRV-Z	SRV-R	BTM	WAL	FIN-Z	FIN-R					
XX	POKT MT	0	5	◆	3	2	0	0					
SNO	TOOL	NOM-ϕ	NO.	APPR-X	APPR-Y	TYPE	ZFD	DEP-Z	WID-R	C-SP	FR	M	M
R1	E-MILL	20.A		?	?	CW	G01	5	12	14	0.135		
FIG	PTN	X	Y	R/θ	I	J	P	CNR					
1	*CW-SH	◆	◆	?	50	50	4	◆					*Note
2	LINE	0	-100										
3	LINE	100	-100										
4	LINE	200	0										
5	PEP-EN	◆	◆	◆	◆	◆	◆	◆					
FIG	PTN	P1X/CX	P1Y/CY	P3X/R	P3Y	CN1	CN2	CN3	CN4				
6	SQR	0	0	100	100								

Note) Depress [SHAPE SHIFT] and [CCW ↺].

Face Machining (POKT VLY)

(Description)

In this program, both external and internal shapes are defined as arbitrary shapes.



<Program Example>

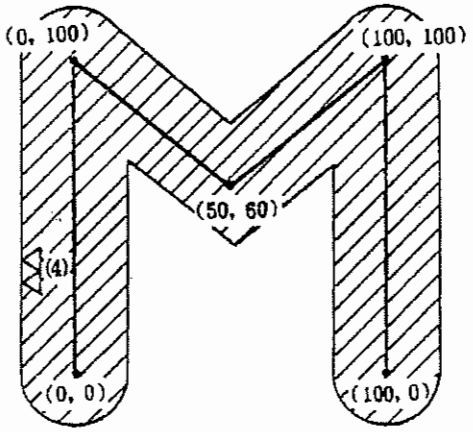
UNO	UNIT	DEP	SRV-Z	SRV-R	BTM	WAL	FIN-Z	FIN-R				
XX	POKT VLY	0	5	◆	5	6	0.42	0.294				
SNO	TOOL	NOM-ϕ	NO.	APPR-X	APPR-Y	TYPE	ZFD	DEP-Z	WID-R	C-SP	FR	M M
R1	E-MILL	20.	A	?	?	CW	G01	4.58	12	14	0.135	
F2	E-MILL	20.	A	?	?	CW	G01		12	14	0.256	
FIG	PTN	X	Y	R/θ	I	J	P	CNR				
*①	CW-SH	◆	◆	100	100	100	4	◆				
2	LINE	100	0									
3	LINE	180	0									
4	CW	200	20	20								
5	LINE	200	100									
6	PEP-EN	◆	◆	◆	◆	◆	◆	◆				
*⑦	LINE	100	60	15.75								
8	LINE	?	?	22.5								
9	LINE	140	100	67.5								
10	LINE	?	?	112.5								
11	LINE	100	140	157.5								
12	LINE	?	?	22.5								
13	LINE	60	100	67.5								
14	LINE	?	?	112.5			*(NOTE)					

Note 1) Circled numbers represent the starting point.

Face Machining (SLOT)

(Description)

An arbitrary shape line-line

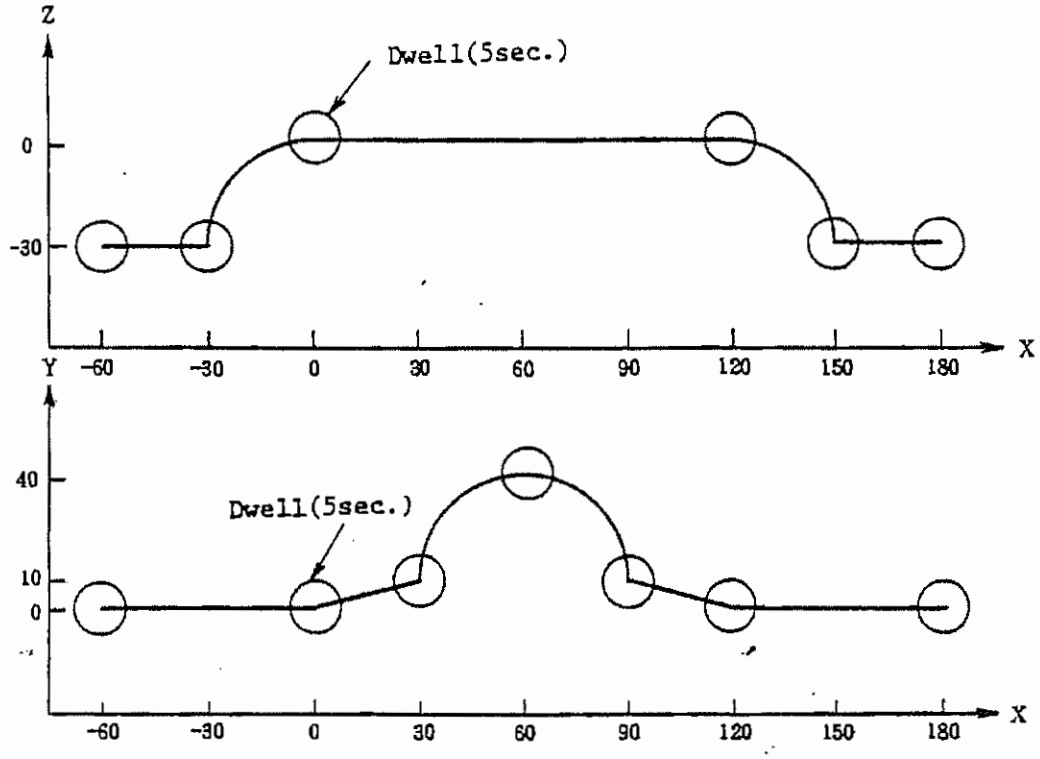


Bottom roughness ▽▽▽ (5)

<Program Example>

UNO	UNIT	DEP	SRV-Z	SLOT-WID	BTM	WAL	FIN-Z	FIN-R			
XX	SLOT	0	5	25	4	5	0.6	0.42			
SNO	TOOL	NOM-ϕ	NO.	APPR-X	APPR-Y	TYPE	ZFD	DEP-Z	WID-R	C-SP	FR	M	M
R1	E-MILL	20.A		?	?	CW	G01	4.4	◆	14	0.135		
F2	E-MILL	20.A		?	?	CW	G01	◆	◆	14	0.32		
FIG	PTN	X	Y	R/θ	I	J	P	CNR					
1	LINE	0	0										
2	LINE	0	100										
3	LINE	50	60										
4	LINE	100	100										
5	LINE	100	0										

Manual Program Mode



<Program Example>

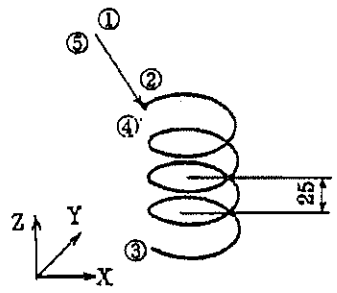
UNO	UNIT	TOOL	NOM-ϕ	NO.						
xx	MANU-PRO	E-MILL	20	A						
SNO	G1	G2	DATA1	DATA2	DATA3	DATA4	DATA5	DATA6	S	M/B
1	0		X-60	Y 0	Z-30				100	
2	1		X-30			F0.01				
3	3	18	X 0		Z 0		R30			
4	4		D 5							
5	1	17	X 30	Y 10						
6	2		X 90	Y 10			R30			
7	1		X 120	Y 0						
8	91									
9	3	18	X 30		Z 30		R30			
10	1	17	X 30							

■ Manual Program Mode(herical cutting)

(Description)

Pitch 25

Arc 360°



UNO	UNIT	TOOL	NOM-Ø	NO.
XX	MANU PRO	E-MILL	20.A	

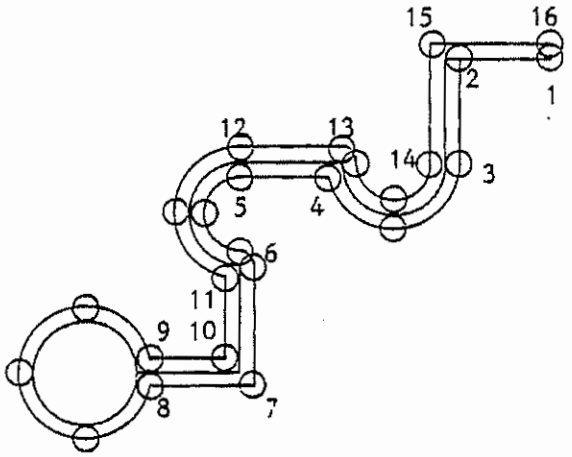
SNO	G1	G2	DATA1	DATA2	DATA3	DATA4	DATA5	DATA6	S	M/B
1	0	94	X0	Y0	Z0				500	M3
2	1		X-100	Y-100	Z-20	F1000				
3	2		X-100	Y-100	Z-120	I50	J0	P25		
4	3	91	X0	Y0	Z100	I50	J0	P25		
5	0		X100	Y100	Z20					

☒ Manual Program Mode(tool diameter compensation)

(Description)

G41 COMPENSATE LEFT

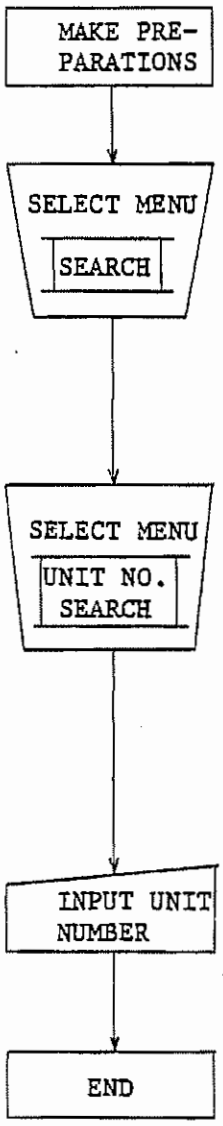
G42 COMPENSATE RIGHT



SNO	G1	G2	DATA1	DATA2	DATA3	DATA4	DATA5	DATA6	S	M/B
1	0	94	X0.	Y0.	Z0.	F500			1000	M3
2	1	41	X-50.							
3	1			Y-50.						
4	2		X-100.		I-25.					
5	1		X-150.							
6	2	91		Y-50.		J-25.				
7	1			Y-50.						
8	1		X-50.	Y0.						
9	2		X0.	Y0.	I-20.					
10	1	90	X-150.							
11	1			Y-100.						
12	3			Y-50.		J-25.				
13	1		X-100.							
14	3	91	X50.		I25.					
15	1			Y50.						
16		90	X0.	Y0.	Z0.					

NOTE: A block without the move command, if any, will cause compensation to be cancelled.

3.2 Unit No. Search Procedures



Ready to prepare program

Depress [MENU SELECT] key to display the following menu and depress the [SEARCH] menu key.

	SEARCH	PROGRAM	LINE INSET	UNIT MOVE	ERASE	DETAILED INFORM	SHAPE COPY	UNIT COPY

Depress

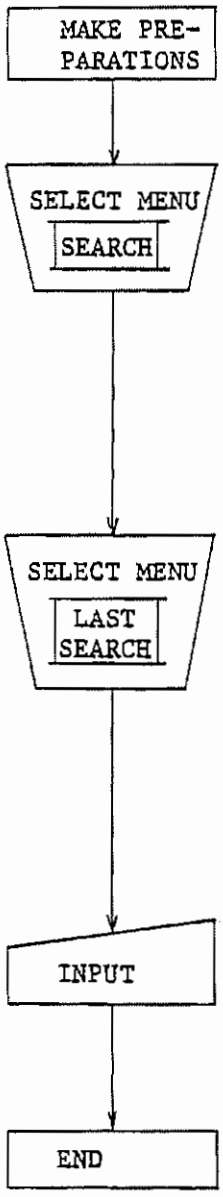
Depress [UNIT NO. SEARCH] menu key.

	UNIT NO. SEARCH	PROGRAM	LAST SEARCH		UNIT SEARCH	TOOL SEARCH		

Depress

Enter the unit No. to be searched, using ten keys. Then, depress the [INPUT] key.

3.3 Program End Search Procedures



Ready to prepare program

Depress the MENU SELECT key to display the following menu.

Depress the SEARCH menu key.

	SEARCH	PROGRAM	LINE INSET	UNIT MOVE	ERASE	DETAILED INFORM	SHAPE COPY	UNIT COPY

Depress

Depress the LAST SEARCH menu key.

	UNIT NO. SEARCH	PROGRAM	LAST SEARCH		UNIT SEARCH	TOOL SEARCH		

Depress

Depress the INPUT key.

3.4 Unit Name Search Procedures

MAKE PRE-PARATIONS

Ready to prepare program.

SELECT MENU
[SEARCH]

Depress the [MENU SELECT] key to display the following menu.
Depress the [SEARCH] key.

	SEARCH	PROGRAM	LINE INSET	UNIT MOVE	ERASE	DETAILED INFORM	SHAPE COPY	UNIT COPY

Depress

SELECT MENU
[UNIT SEARCH]

Depress the [UNIT SEARCH] menu key.

	UNIT NO. SEARCH	PROGRAM	LAST SEARCH		UNIT SEARCH	TOOL SEARCH		

Depress

SELECT MENU
[ ]

Select the unit name from the following menu and depress that key.

POINT MACH-ING	LINE MACH-ING	FACE MACH-ING	MANUAL PROGRAM	OTHER	WPC	OFFSET	END	GROUP CHECK

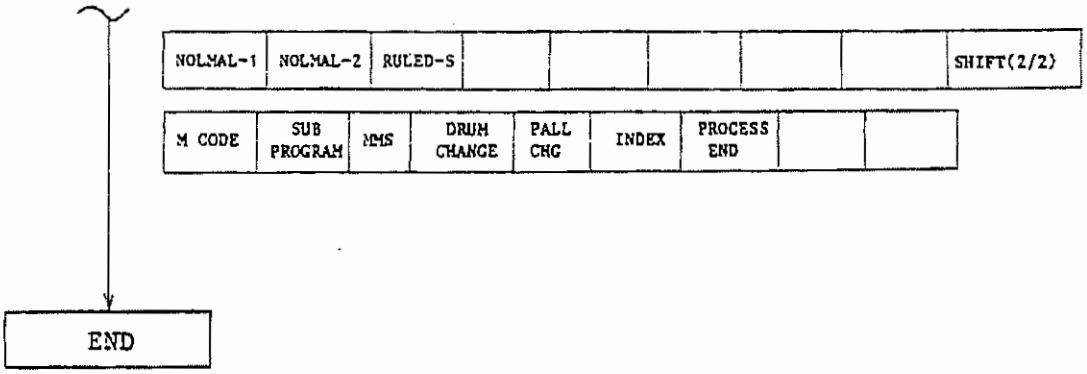
DRILLING	RGH CBOR	RGH BCB	REAMING	TAPPING	BORING	BK-CBORE	CIRC MIL	CBOR-TAP

BORING T1	BORING S1	BORING T2	BORING S2					

LINE CTR	LINE RGT	LINE LFT	LINE OUT	LINE IN	CHMF RGT	CHMF LFT	CHMF OUT	CHMF IN

FACE MIL	TOP EMILL	STEP	POCKET	PCKT MT	PCKT VLY	SLOT	3-D SURFACE	

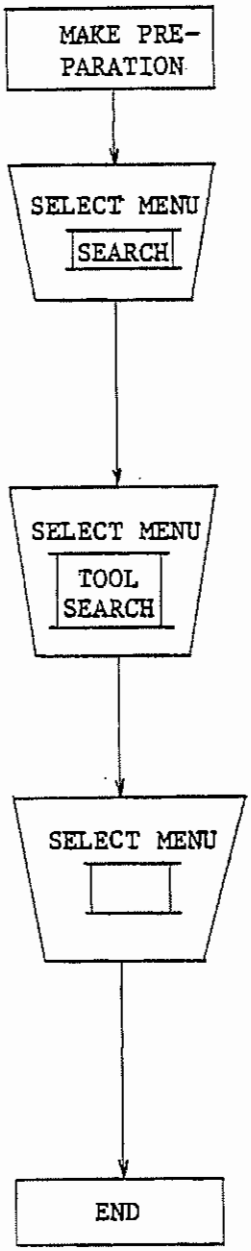
ROTETE1	ROTETE2	ROTETE3	ROTETE4	PARALL.1	PARALL.2	PARALL.3	PARALL.4	SHIFT(1/2)



NOTE 1: The menu GROUP CHECK is invalid here.

NOTE 2: Menus associated with options are valid whether or not options are employed.

3.5 Tool Name Search Procedures



Ready to prepare program.

Depress the MENU SELECT key to display the following menu.

Depress the SEARCH menu key.

	SEARCH	PROGRAM	LINE INSET	UNIT MOVE	ERASE	DETAILED INFORM	SHAPE COPY	UNIT COPY

Depress

Depress the TOOL SEARCH menu key.

	UNIT NO. SEARCH	PROGRAM	LAST SEARCH		UNIT SEARCH	TOOL SEARCH		

Depress

Select the tool name from the following menu and depress the key.

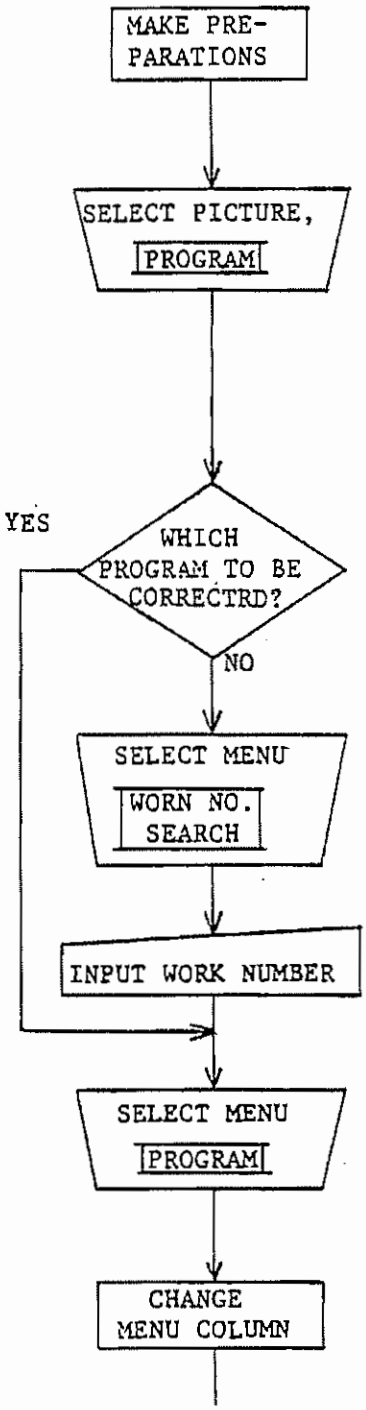
ENDMILL	FACEMILL	CHAMF. CUTTER	BALL ENDMILL	OTHER TOOL	TOUCH SENSOR		NO TOOL	NEXT (1/2)

CENTER DRILL	DRILL	BACKSPOT FACER	REAMER	TAP	BOHRING BAR	BACK BOR. BAR	CHIP VACUUM	NEXT (2/2)

NOTE 1: NO TOOL menu is invalid here.

NOTE 2: For a tapping tool, the METRIC, UNFY, PIPE, etc. must be specified.

3.6 Procedure for Inserting (Erasing) one line:



Put the PROGRAM switch (with a key) in the ENABLE position, then depress the RESET key.

Depress the DISPLAY SELECT key to display the menu shown below. Then, depress the PROGRAM (menu key).

POSITION	COMMAND	GRAPHIC	PROGRAM	TOOL FILE	TOOL DATA	TOOL LAYOUT	PAR	DIAGNOS.

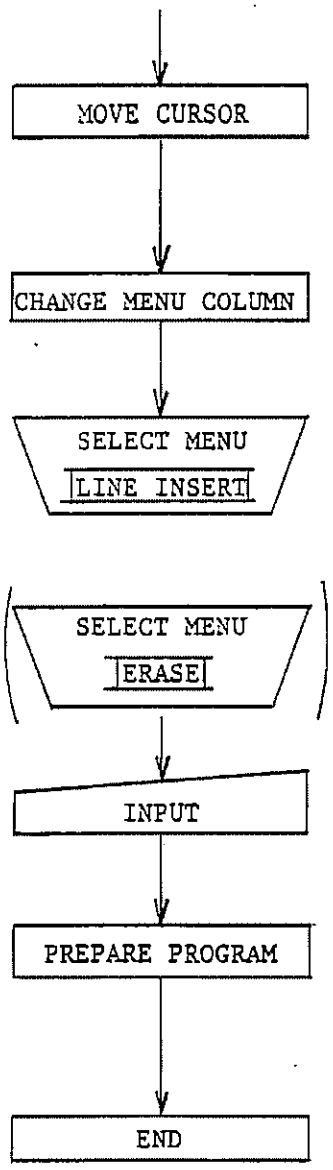
Is a program to be corrected displayed on the picture?

WORK NO. SEARCH	UNIT NO. SEARCH	PROGRAM				DETAILED INFORM.	WPC MSR	PROGRAM FILE

Input a workpiece number to be corrected.

WORK NO. SEARCH	UNIT NO. SEARCH	PROGRAM				DETAILED INFORM.	WPC MSR	PROGRAM FILE

Depress the MENU SELECT key.



Depress the CURSOR key and move the cursor to the line to be inserted (erased).

Depress the MENU SELECT key.

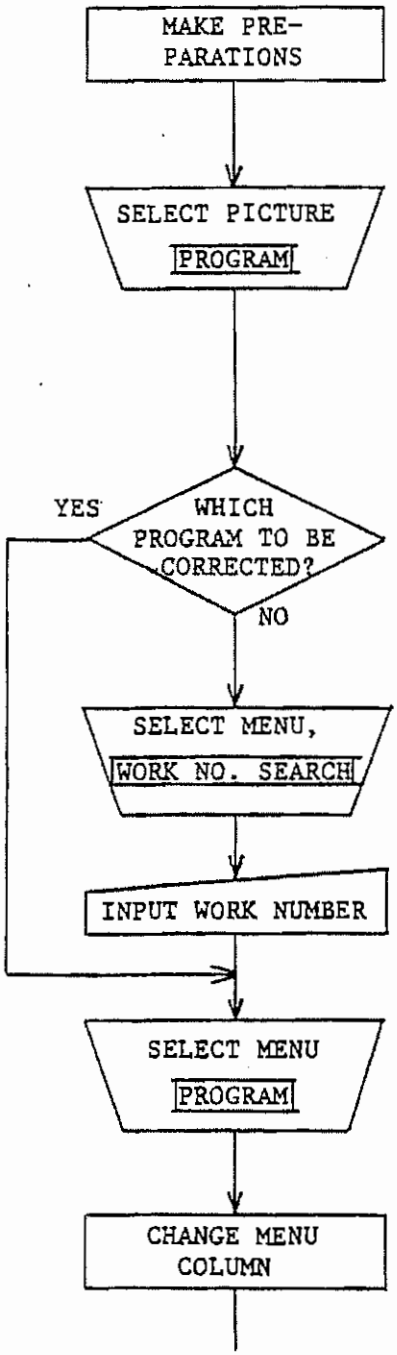
	SEARCH	PROGRAM	LINE INSERT	UNIT MOVE	ERASE	DETAILED INFORM.	SHAPE COPY	UNIT COPY
			↑		(↑)			

Depress the INPUT key.

Prepare the program to be inserted.
(This is not related to erasing.)

Note) Insertion in a unit is impossible.

3.7 Procedure for Moving a Unit



Put the PROGRAM switch (with a key) in the ENABLE position, then depress the RESET key.

Depress the DISPLAY SELECT key to display the menu shown below. Then, depress the PROGRAM (menu key).

POSITION	COMMAND	GRAPHIC	PROGRAM	TOOL FILE	TOOL DATA	TOOL LAYOUT	PAR	DIAGNOS.

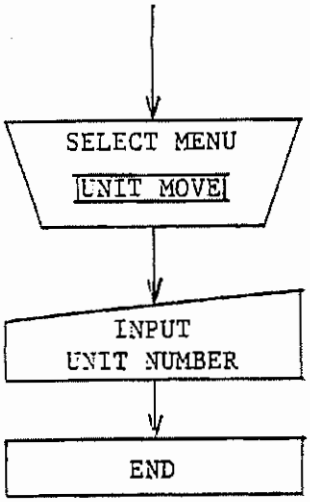
Has the program to be corrected been displayed on the screen?

WORK NO. SEARCH	UNIT NO. SEARCH	PROGRAM				DETAILED INFORM.	WPC MSR	PROGRAM FILE

Input a workpiece number to be corrected.

WORK NO. SEARCH	UNIT NO. SEARCH	PROGRAM				DETAILED INFORM.	WPC MSR	PROGRAM FILE

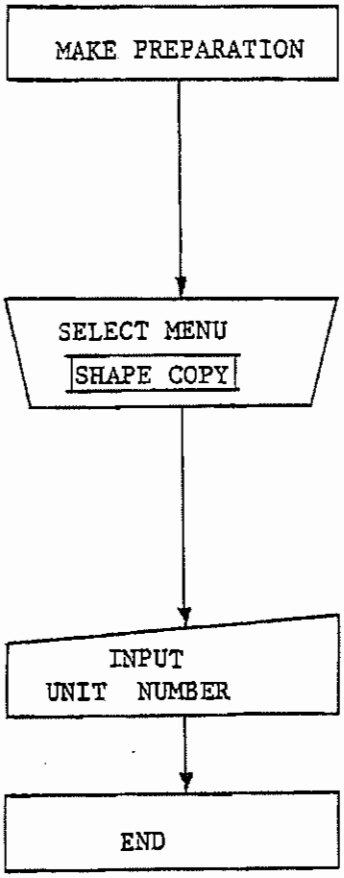
Depress the MENU SELECT key.



	SEARCH	PROGRAM	LINE INSERT	UNIT MOVE	ERASE	DETAILED INFORM.	SHAPE COPY	UNIT COPY

Input the unit number to be moved.

3.8 Procedure for Copying a Shape



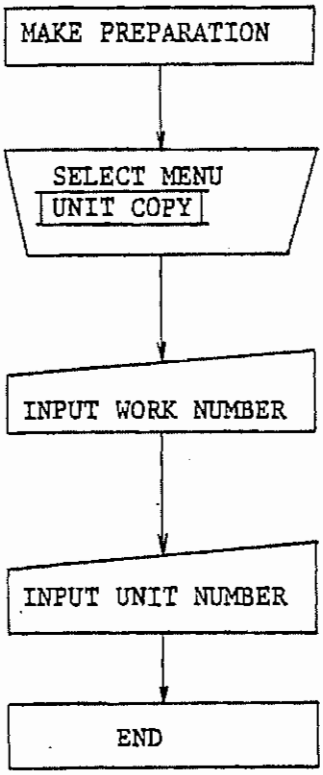
While a program is being prepared, the cursor has entered in PTN at the head of FIG in point, line or face machining. Thus, the system is waiting for an input. Or with the cursor placed at G1 at the head of SNO in the manual program unit, the system is waiting for an input.

Depress the MENU SELECT key and display the menu column below. Then, depress the SHAPE COPY.

	SEARCH	PROGRAM	LINE INSERT	UNIT MOVE	ERASE	DETAILED INFORM.	SHAPE COPY	UNIT COPY

Input a unit number in which a shape to be copied is available.

3.9 Procedure for Copying a Unit



Although a program is being prepared, the screen is ready for input of a new unit.

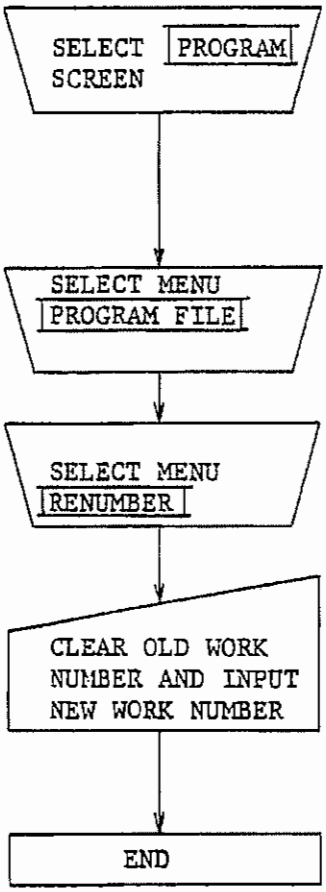
Push the MENU SELECT key to call the following menu columns. Then, push UNIT COPY.

	SEARCH	PROGRAM	LINE INSERT	UNIT MOVE	ERASE	DETAILED INFORM.	SHAPE COPY	UNIT COPY

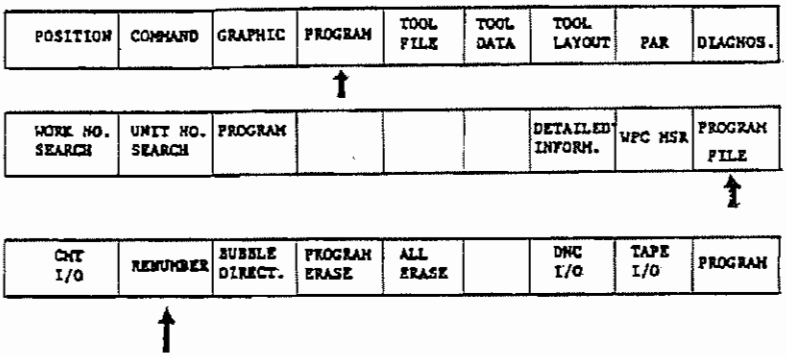
Input a work number of the program which has the unit to be copied.

Input a unit number for the unit to be copied.

3.10 Procedure for Changing a Work Number



Push the DISPLAY SELECT key to display the menu shown below. Then, push the PROGRAM.

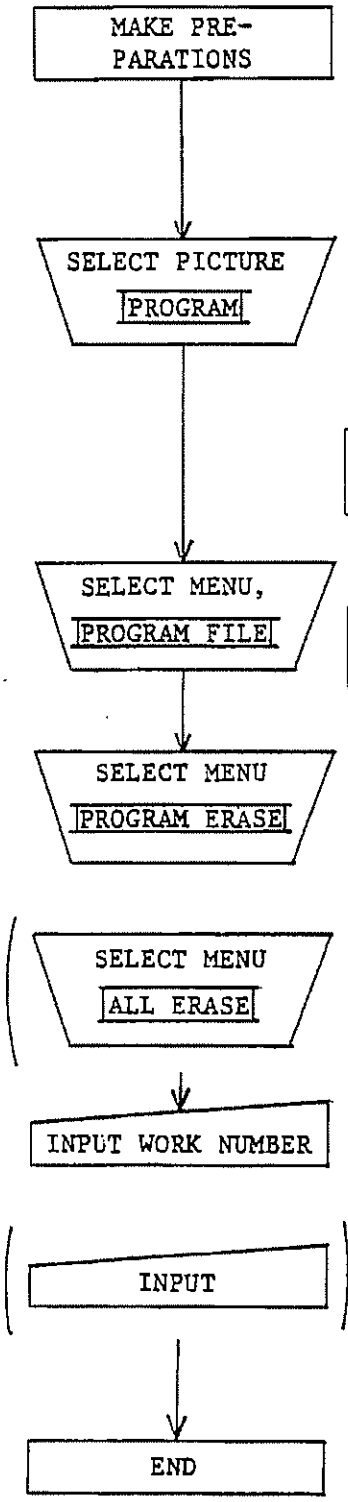


Push [.] for old work number and push [INPUT] for new work number.

Example: Changing the work number 99 into 100

After inverting the menu by pushing [RENUMBER] menu key, push [9], [9], [.], [1], [0], [0] and [INPUT] .

3.11 Procedure for Erasing Programs (all programs)



Throw the PROGRAM switch (with a key) to the ENABLE position. Then, depress the RESET key.

Depress the DISPLAY SELECT key to display the menu shown below. Then, depress the PROGRAM (menu key).

POSITION	COMMAND	GRAPHIC	PROGRAM	TOOL FILE	TOOL DATA	TOOL LAYOUT	PAR	DIAGNOS.

WORK NO. SEARCH	UNIT NO. SEARCH	PROGRAM				DETAILED INFORM.	WPC MSR	PROGRAM FILE

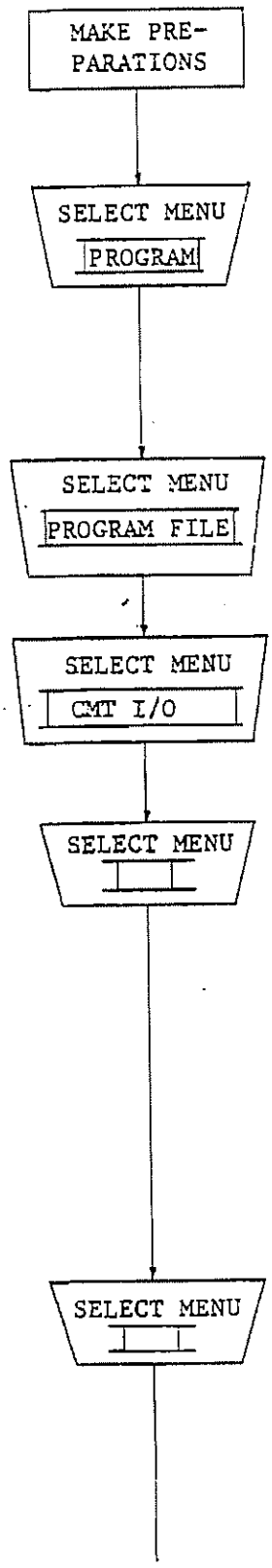
CNT I/O	RENUMBER	BUBBLE DIRECT.	PROGRAM ERASE	ALL ERASE		DNC I/O	TAPE I/O	PROGRAM

To erase all programs

Input a workpiece number to be erased.

Merely depress the INPUT key (to erase all programs).

3.12 Procedure for Using the CMT Interface



Connect the cable between the cassette deck and the NC unit. Switch on the cassette deck.
Load a cassette tape.

Depress the DISPLAY SELECT key and display the menu shown below, then depress the PROGRAM (menu key).

POSITION	COMMAND	GRAPHIC	PROGRAM	TOOL FILE	TOOL DATA	TOOL LAYAUT	PAR	DIAGNOS.

WORK NO. SEARCH	UNIT NO. SEARCH	PROGRAM				DETAILED INFORM.	WPC MSR	PROGRAM FILE

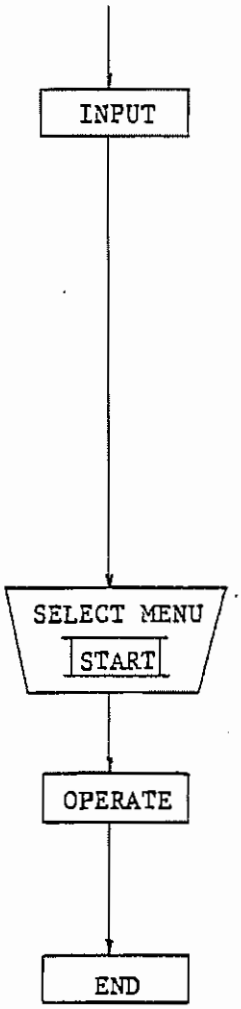
CMT I/O	RENUMBER	BUBBLE DIRECT.	PROGRAM ERASE	ALL ERASE		DNC I/O	TAPE I/O	PROGRAM

LOAD CMT→NC	SAVE NC→CMT	COMPARE NC=CMT	PROGRAM ALL LOAD					

1. LOAD CMT→NC — puts cassette tape information in the NC unit.
2. SAVE NC→CMT — saves NC information on the cassette tape.
3. COMPARE NC=CNT — checks that the NC information is the same as the cassette tape information.
4. PROGRAM ALL LOAD — puts all the program information only on the cassette tape in the NC unit.

PROGRAM	TOOL DATA	TOOL FILE	PAR	TOOL OFFSET	WORK OFFSET	ERASE		START

Select either PROGRAM, TOOL DATA, TOOL FILE, PAR, TOOL OFFSET or WORK OFFSET. Depress the menu key and the cursor will move. These are unnecessary in the case of PROGRAM ALL LOAD .

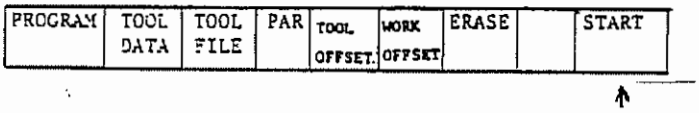


(1) PROGRAM (16 sets)
Input a work number and depress the [INPUT] key.

(2) TOOL DATA (4 sets)
Input a drum number and depress the [INPUT] key. * See note below.

(3) TOOL FILE, PAR, TOOL OFFSET and WORK OFFSET (1 set each). Depress [1] and [INPUT].

(4) This is unnecessary in the case of [PROGRAM ALL LOAD].

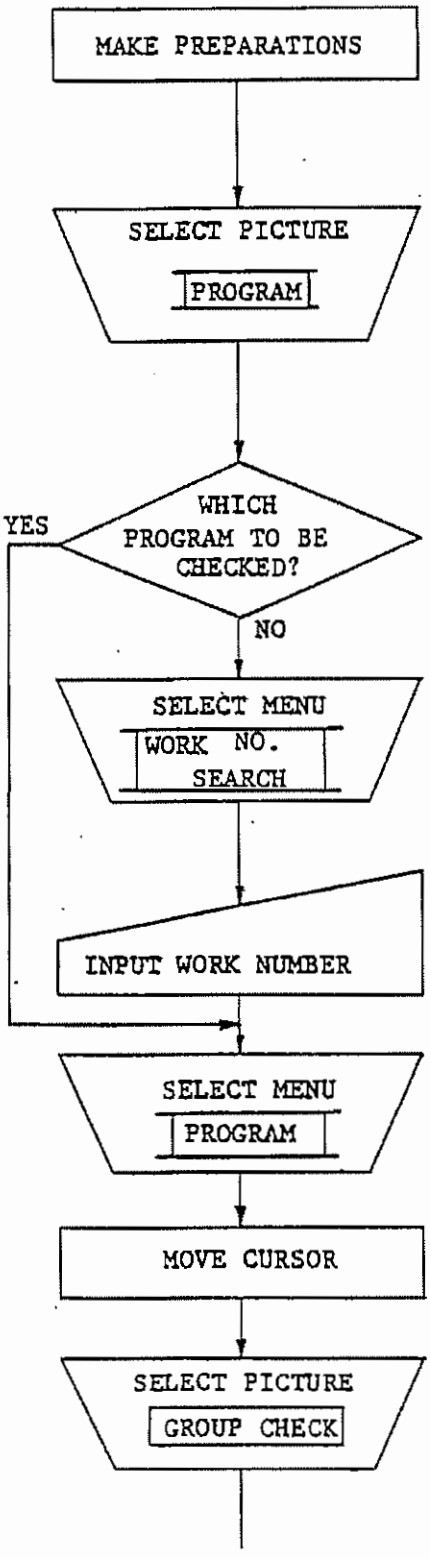


During the operation, menu key [START] is displayed inversed without going out.
At the same time when the program has ended, the [START] menu key dieplayed as inverted will go out.

NOTE : In the case of a drum changer, input a drum number.

3.13 Group Check

Depress this menu to check a shape in basic coordinates while preparing a program.



Throw the PROGRAM switch (with a key) to the ENABLE position. And depress the [RESET] key.

Depress the [DISPLAY SELECT] key and display the menu below. Then, depress the [PROGRAM] (menu key).

POSITION	COMMAND	GRAPHIC	PROGRAM	TOOL FILE	TOOL DATA	TOOL LAYAUT	PAR	DIAGNOS.

Has the program to be checked been displayed on the picture?

Depress menu key [WORK NO. SEARCH].

WORK NO. SEARCH	UNIT NO. SEARCH	PROGRAM				DETAILED INFORM.	WPC MSR	PROGRAM FILE

Input a workpiece number to be checked.

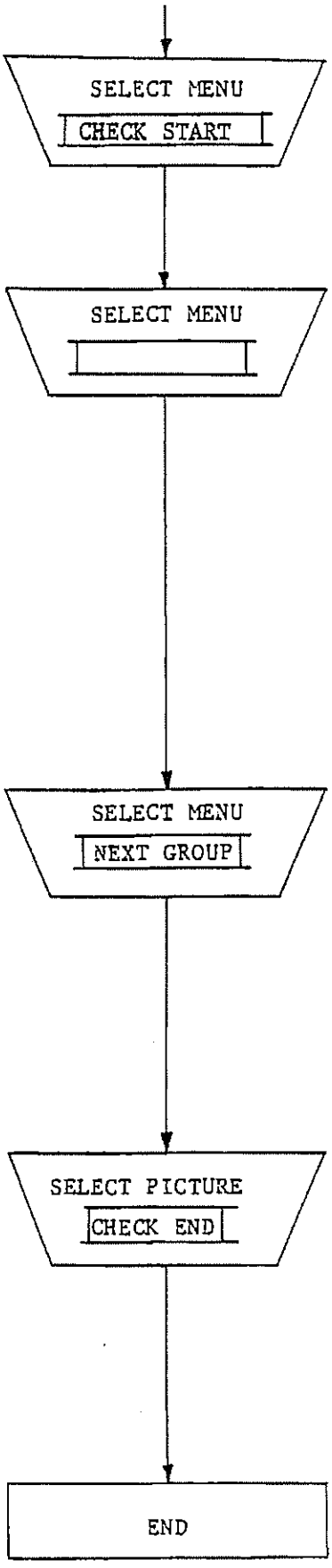
Depress menu key [PROGRAM].

WORK NO. SEARCH	UNIT NO. SEARCH	PROGRAM				DETAILED INFORM.	WPC MSR	PROGRAM FILE

Depress the CURSOR key and move the cursor to the machining unit position at which the following menu is available.

Depress menu key [GROUP CHECK].

POINT MACH-ING	LINE MACH-ING	FACE MACH-ING	MANUAL PROGRAM	OTHER	WPC	OFFSET	END	GROUP CHECK



Depress menu key [CHECK START].

CHECK START	NEXT GROUP	CHECK STEP			SHAPE ERASE	DISPLAY MODE	COORD OFFSET	CHECK END

Shapes from the hed of the program to the subsequent basic coordinates are displayed on the picture.

CHECK START	NEXT GROUP	CHECK STEP			SHAPE ERASE	DISPLAY MODE	COORD OFFSET	CHECK END

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

(1) [CHECK STEP] Every time when this key is depressed, a shape is displayed in steps.

(2) [SHAPE ERASE] Depressing this key will erase the shape being displayed.

(3) [DISPLAY MODE] Depressing this key will permit selecting a plane displayed. (See "Graphic Display" in Operating Manual.)

(4) [COORD OFFSET] Depressing this key will permit changing a display viewpoint. (See "Graphic Display" in Operating Manual.)

Depress menu key [NEXT GROUP].

CHECK START	NEXT GROUP	CHECK STEP			SHAPE ERASE	DISPLAY MODE	COORD OFFSET	CHECK END

Shapes from the next basic coordinates to those after next are displayed on the picture. Every time when this menu key is depressed, shapes can be checked one after another in basic coordinates.
The system will not operate unless the following basic coordinates are available.

Depress menu key [CHECK END].

CHECK START	NEXT GROUP	CHECK STEP			SHAPE ERASE	DISPLAY MODE	COORD OFFSET	CHECK END

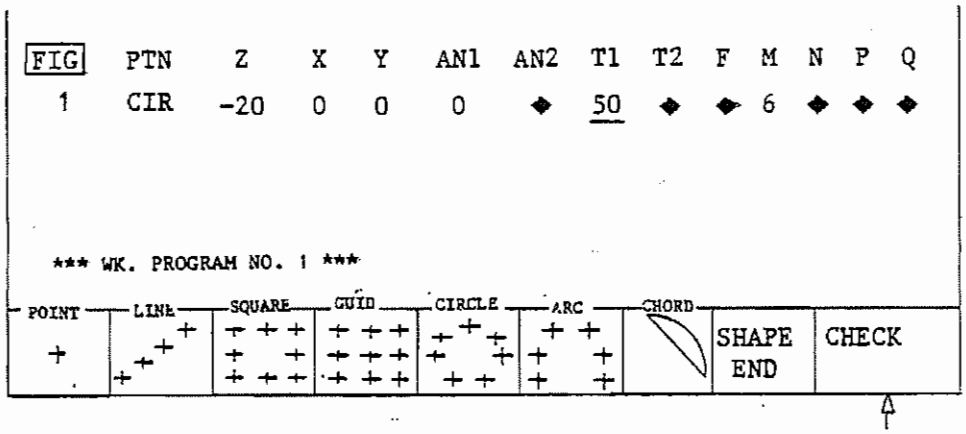
Then, the display will return to the status before [GROUP CHECK] on the program picture is depressed.

3.14 Unit Check

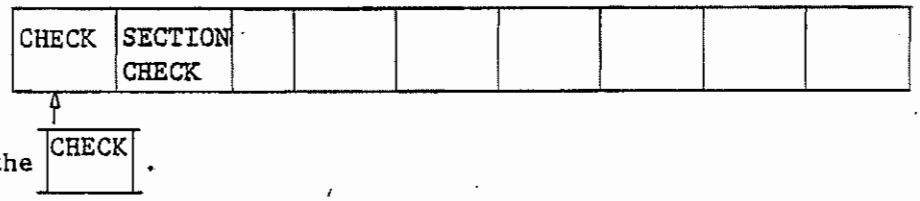
(i) Checking a Unit → CHECK

The CHECK menu key is located at the end of the POINT, LINE and FACE SHAPE input menu array. Depressing the CHECK key entering each shape will change the program picture over to the CHECK picture. And the shape will be drawn on the X, Y picture. (In the case of point machining, it is necessary to depress the CHECK and then the CHECK too.

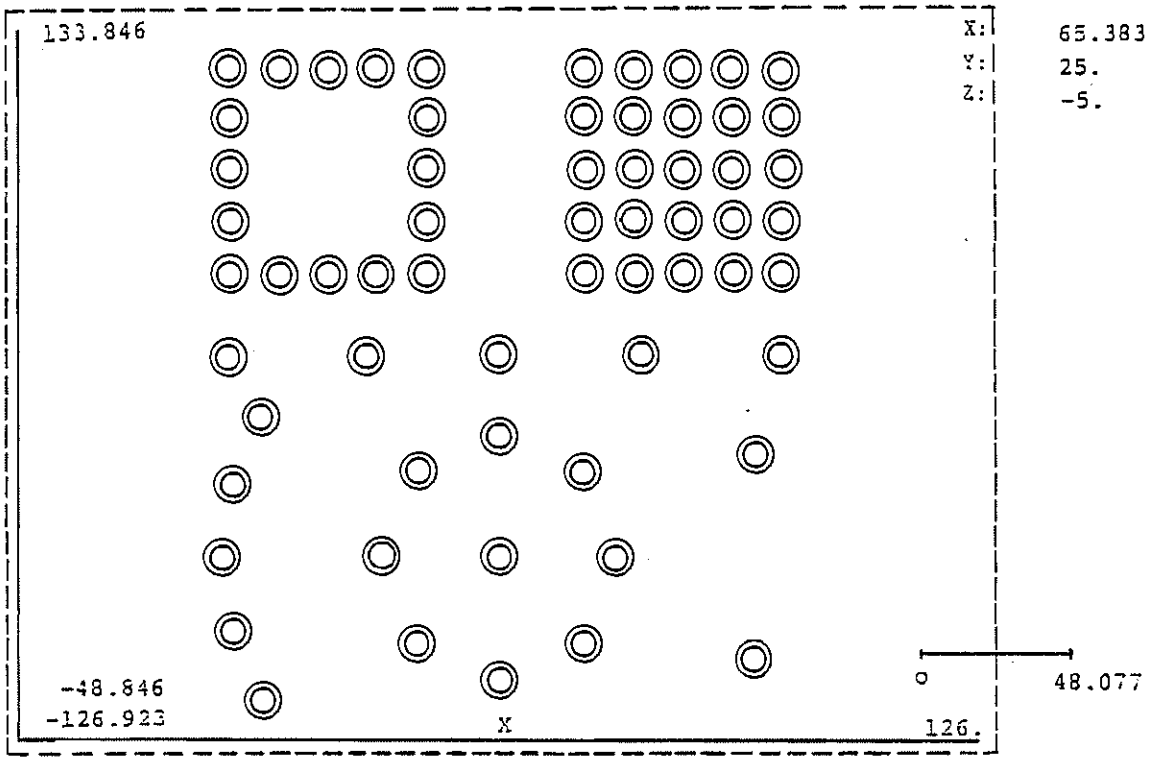
(Example)



CHECK CONTINUE will cause the display to draw a spot machining shape as illustrated below.



Depress the CHECK.



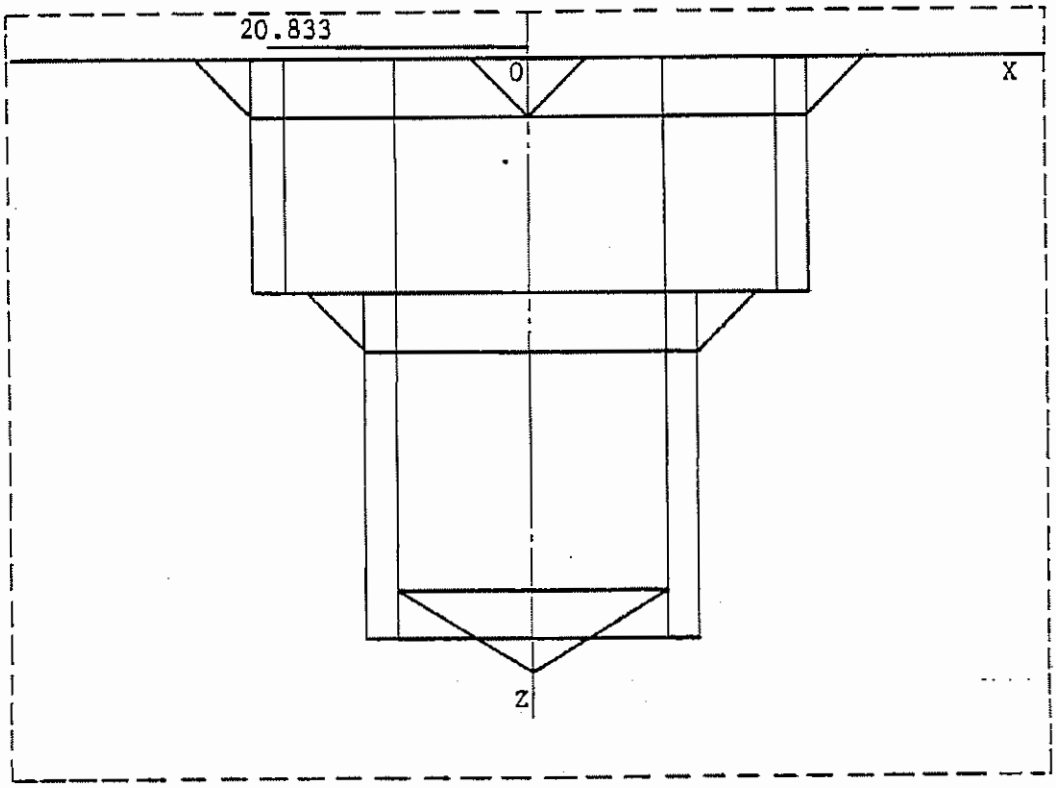
In the automatic mode, depress CHECK and the scale will be automatically set so that the graph defined in the program will appear to the full size of the picture.

CHECK END will cause a return to the original picture.

(ii) Checking a Unit + SECTION CHECK

SECTION CHECK displays the development of tools for point machining. Only in the automatic mode and for XZ coordinates. Depress the CHECK, last key of point machining menu, and menu CHECK and SECTION CHECK will be displayed. Then depress SECTION CHECK key and the same menu as CHECK will be displayed.

Depress CHECK CONTINUE so that spot machining unit tools will develop by turns on the picture.



WNO. 9999

UNO. 1 BORE S2

	HOLE-ϕ	HOLE-DEP
SPOT	10.	
DRILL	24.	45.794
E-MILL	24.	50.
E-MILL	44.	20.
BORING	30.	50.
BORING	50.	20.
CHF-M	5.	
CHF-M	5.	

Colors change from process (tool) to process (tool).

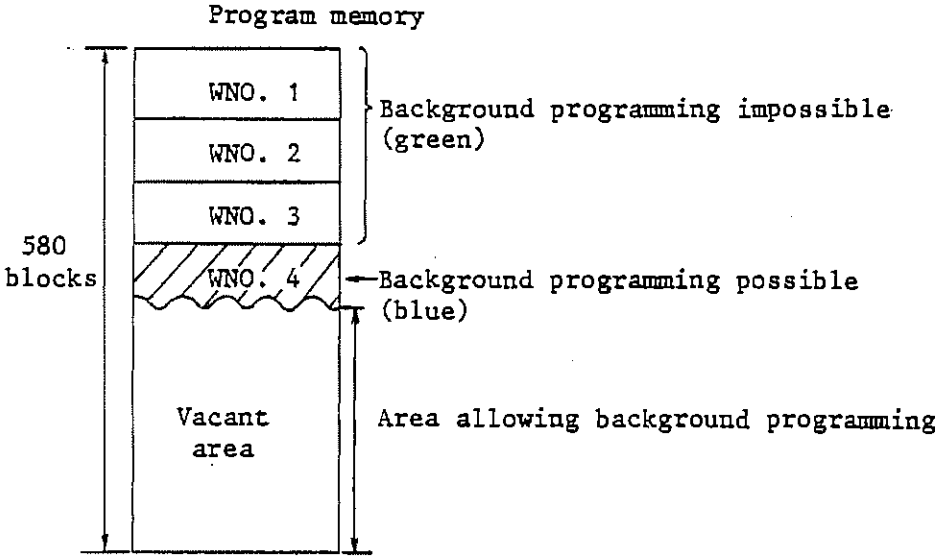
Depress CHECK STEP and the display appears for each tool.

3.15 Background Programming

Background programming means the preparation or edition of another program when machining is being executed according to a program. To avoid adverse effect on the program executing the machining, the following limitations must be observed:

i) A background program to be prepared should not be the program currently executing the machining. It should be either the lowest program belonging to the program memory(Note 1) or an entirely new program (Note 2).

Example:



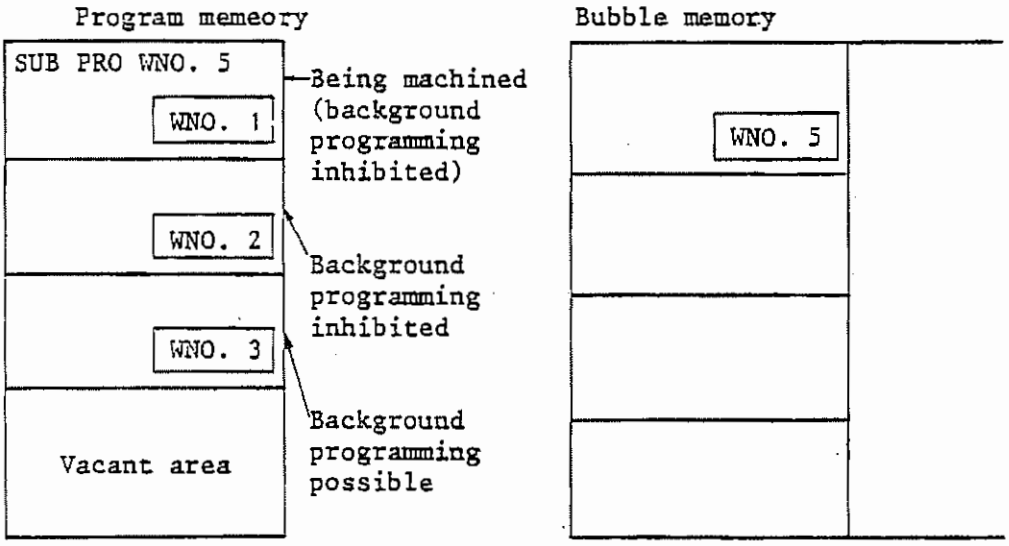
ii) The capacity in which background programming is possible is equivalent to that of the vacant area of the program memory.(See the figure above.) For example,if the vacant area of the RAM is 10 blocks, the capacity of background programming will be also 10 blocks.

Note 1) The program memory means memory area to hold programs in the NC.

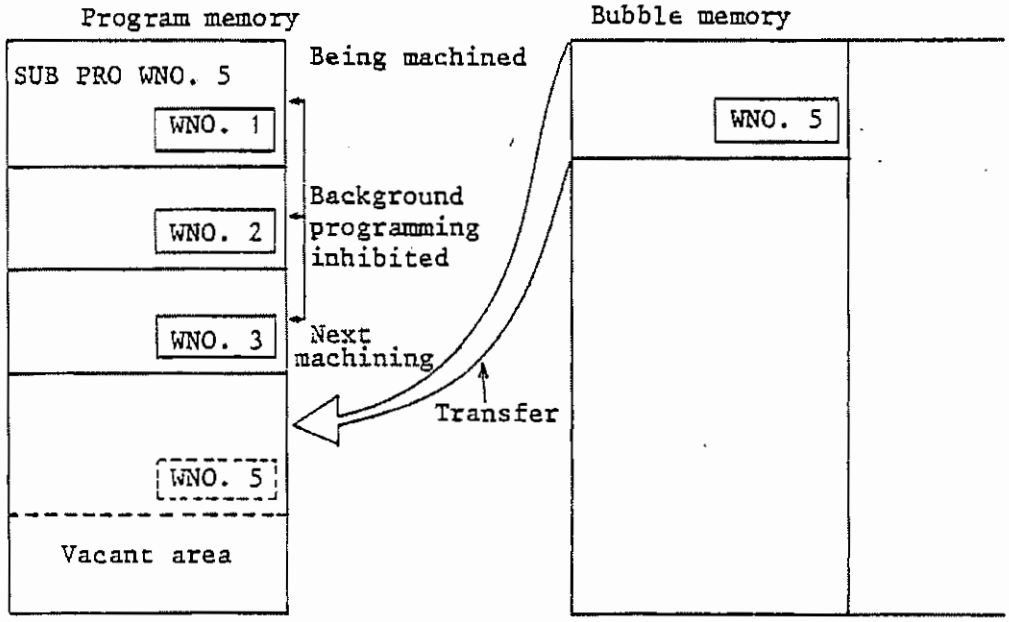
2) A program whose work No. is blue on the program management display allows background programming.
If work No. is green, that program can not be prepared as background program.

iii) When using a bubble memory, the bubble memory and program memory will automatically exchange (transfer) programs between them in the course of machining if sub-programs for the main program currently engaged in machining are located in the bubble memory. In this while, the status of the program memory is so unstable that background programming is inhibited. That is, background programming is inhibited by the alarm which is given immediately when programs are exchanged (transfered) between the bubble memory and program memory during background programming.

(Before transfer)



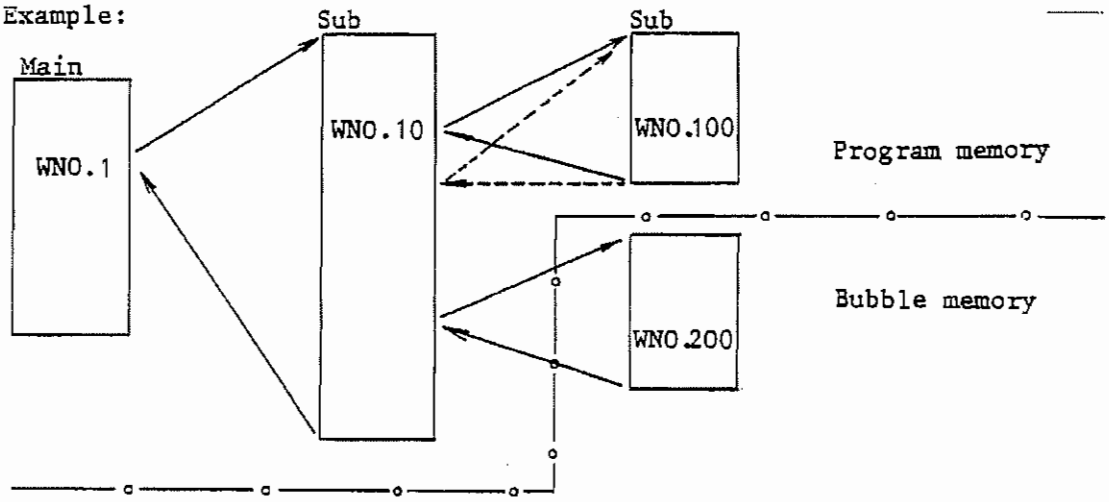
(During transfer)



Supplement

1. In case the size of the entire machining programs does not exceed 580 blocks, no programs will be transferred between the bubble and program memories, if machining is started after transferring all the machining programs to the program memory. In this case, background programming is not inhibited at all because the memories are stable.
2. In case the size of the entire machining programs exceeds 580 blocks, as many machining programs as possible should be transferred to the program memory by WORK NO.SERCH before starting machining so that the machining time until starting of transfer of machining programs between the bubble and program memories may become the longest. In this status, background programming will be possible so long as no programs are exchanged between the bubble memory and program memory.

Example:



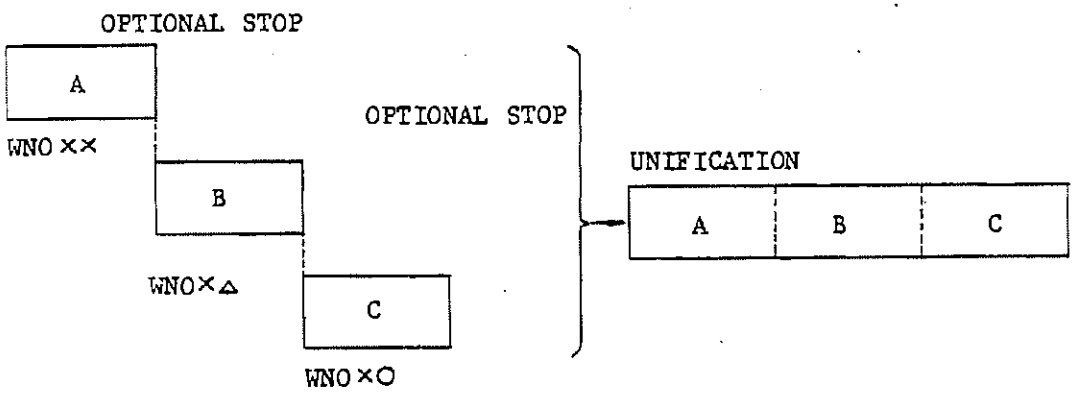
In this case, no program is transferred between the bubble memory and RAM until the WNO. 200 sub-program is called.

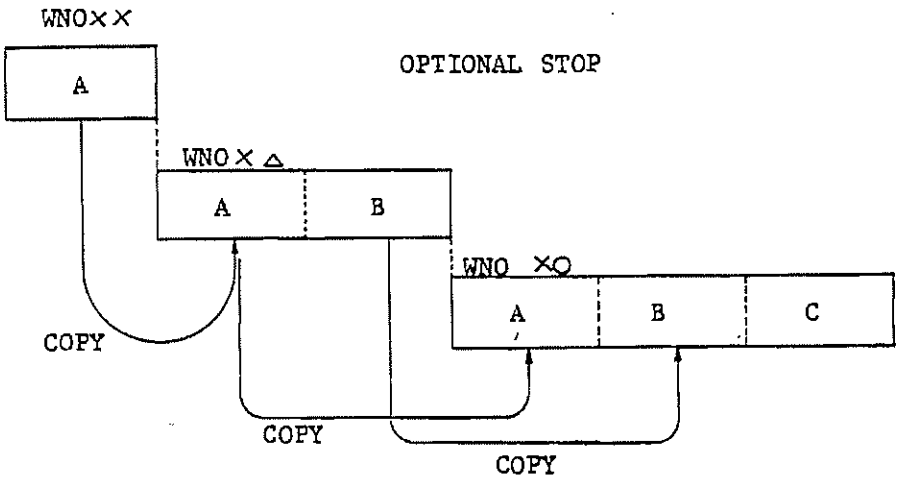
If M01 (optional stop) prepared with the M code unit is provided in front of the sub-program unit used for calling a sub-program to be transferred from the bubble memory before starting machining, background programming will not be inhibited abruptly because the optional stop is effective.

After optional stop, machine operation or programming may begin. When machine starts operating, continue programming by one of the following process.

a) Continue programming but under a different program number. When completed, under a third program number, copy the previous 2 programs.

b) Under a different program number, copy the program data and continue programming until completed.





3. Programs in the bubble memory cannot be displayed for the purpose of preparing a background programming.

4. Systems with bubble memory cannot perform shape check of a program during background programming.

4. MAZATROL M - 2 COLLECTION OF PROGRAM EXAMPLES

In the text, the following should be taken for engagements:

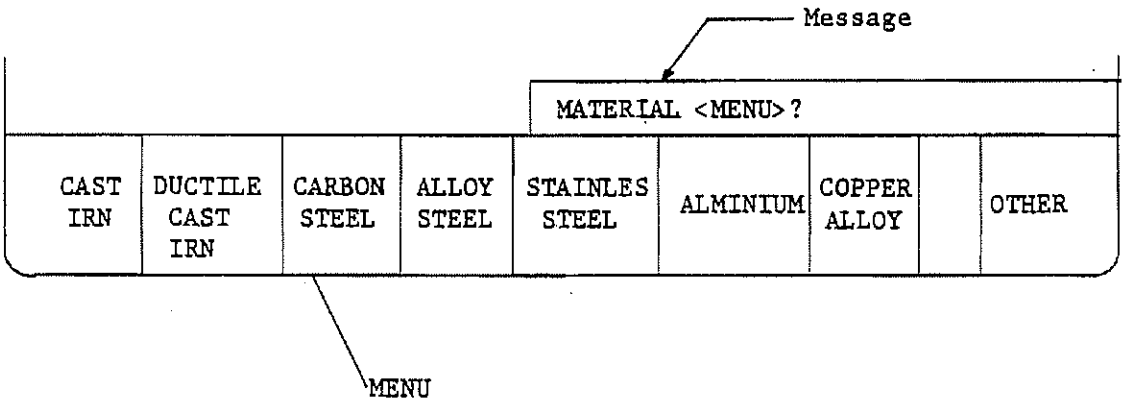
(1) [ ] represents an operating key. (a push button on the control panel)

[INPUT], for example, signifies that the [INPUT] key be depressed.

(2) [ ] represents a menu. The term, menu, means the nine selectors displayed in the lower part of the picture.

[PROGRAM], for example, signifies that the yellow [ ] displayed under the [PROGRAM] be depressed out of the menu selectors displayed in the lower part of the picture.

(3) The term, message, means a question displayed in the lower part of the picture.



(4) The term, cursor, means an oblong frame flashing on the picture. The position at which the cursor is placed is the point at which data are to be set.

In the present NC system, it is necessary to register applicable tools (tools in hand) before preparing a program.

The tools to be registered are four types, i.e., E-MILL, F-MILL, CHAMF cutters and BALL ENDMILL.
Registering tools beforehand is called "TOOL FILE"

In this example, therefore, file tools first of all after switching on the machine.

(1) How to switch on:
Depress the [POWER] in the upper left of the control panel.

Next, validify the program rewriting key-switch. (If it remains invalid, no program can be erased or rewritten.)

(2) TOOL FILE:
To file tools, change the display over to the TOOL FILE picture.

[DISPLAY SELECT] , [TOOL FILE]

Thus, the TOOL FILE picture has appeared. Depressing the [↓] will cause the cursor to appear at (1) . Entering data will sequentially move the cursor to the right.

Carry out operations in accordance with the program sheet given on the next page.

Operation (1) thru (5) will cause the picture to display:

"F-MILL 100A ◆ CBD 4 6 ◆ ."

Likewise, operations (6) thru (10) will cause the picture to display:

"F-MILL 100B ◆ CBD 0.8 6 ◆ ."

Subsequently, set data from No. 3 and on similarly.

TOOL FILE Picture upon Completion

NO.	TOOL	NOM-Ø	MIN-Ø	MAT	DEPTH	NO.	ANG	No.	TOOL	NOM-Ø	MIN-Ø	MAT	DEPTH	NO.	ANG
1	① F-MILL	② 100A	✦	③ CBD	④ 4	⑤ 6	✦	17							
2	⑥ F-MILL	⑦ 100B	✦	⑧ CBD	⑨ 0.8	⑩ 6	✦	18							
3	⑪ E-MILL	⑫ 28A	✦	⑬ HSS	22	2	✦	19							
4	⑭ CHF-M	25	⑮ 17	HSS	◆	2	⑯ 45	20							
5								21							
6								22							
7								23							
8								24							
9								25							
10								26							
11								27							
12								28							
13								29							
14								30							
15								31							
16								32							

NOTE: (NO.) is not actually displayed in the picture. This number represents a program sheet number. That is, if the cursor is positioned there, carry out the operation corresponding to that number and data will be displayed.

The same applies to any picture subsequently displayed.

NOTE: Because the tool file contains very important data on programming and tool layout, automatic determinants for the program must be newly set and tool layout must be newly executed if any data registered on this picture is changed.

NO.	PROGRAM MESSAGE	OPERATION	MEANING OF OPERATION
1	WHICH TYPE OF TOOL <MENU>?	FACEMILL	Select a tool name.
2	NOMINAL DIAMETER ?	1 , 0 , 0 , INPUT	Enter a nominal diameter.
2'	TOOL ID CODE <CODE> ?	A	Enter a suffix (to identify tools with an identical diameter). (Select menu)
3	TOOL MATERIAL <MENU> ?	CARBIDE	Enter a material of the tool (Select menu)
4	MAX. DEPTH OF CUT ?	4 , INPUT	Enter a maximum Z-axial cutting allowance.
5	NUMBER OF TEETH ?	6 , INPUT	Enter the number of teeth.
6	WHICH TYPE OF TOOL <MENU> ?	FACEMILL	See 1.
7	NOMINAL DIAMETER ?	1 , 0 , 0 , INPUT	See 2.
7'	TOOL ID CODE <CODE> ?	B	See 2'.
8	TOOL MATERIAL <MENU>?	CARBIDE	Enter a material of the tool.

NO.	PROGRAM MESSAGE	OPERATION	MEANING OF OPERATION
9	MAX. DEPTH OF CUT ?	[.], [8], [INPUT]	See 4.
10	NUMBER OF TEETH ?	[6], [INPUT]	See 5.
11	WHICH TYPE OF TOOL <MENU> ?	[ENDMILL]	
12	NOMINAL DIAMETER ?	[2], [8], [INPUT]	
12'	TOOL ID CODE <MENU> ?	[A]	
13	TOOL MATERIAL <MENU> ?	[HSS]	
	= do. =		
14	WHICH TYPE OF TOOL <MENU> ?	[CHAMF CUTTER]	
15	(NOMINAL DIAMETER omitted) MINIMUM DIAMETER ?	[1], [7], [INPUT]	Enter the minimum diameter of a chamfering cutter.

NO.	PROGRAM MESSAGE	OPERATION	MEANING OF OPERATION
16	MAT (material) ∿NO (number of teeth) omitted) CUTTER ANGLE ?	[4], [5], [INPUT]	Enter a chamfering cutter cutting edge angle (in degrees)

o Program Examples

Three program examples shown below are provided. They have their respective meanings and become more difficult in the order of No. 1 thru No. 3.

No. 1: Basic Shape

- o With machining classified by pattern, all necessary tools are determined by selecting a pattern.
- o A face milling path is also determined automatically with TYPE and SHAPE.
- o A feed circumferential speed and a feed are automatically determined by selecting materials of the work and of the cutting edge.
- o Drilling is also facilitated because it is classified by pattern.

No. 2:

- o For tapping, up to the lower drill is automatically determined.
- o An irregular drilling pattern can be easily treated too.
- o The cycle of a drill is also automatically determined according to a drill hole depth.

No. 3:

- o According to the roughness of a face to be machined, rough finishing is carried out. Both depth and width to be cut off are automatically determined.
- o Any shape that cannot be represented by a machining pattern can be freely added.
- o For one machining plural shape patterns can be entered. However, there is no tool path. (Jigs and tools can be checked for possible interference.)
- o Up to the lower hole to be bored (BOR-B) is automatically determined.
- o Chamfering can be carried out easily too.

Examples have been selected so that the above-mentioned features can be easily gathered.

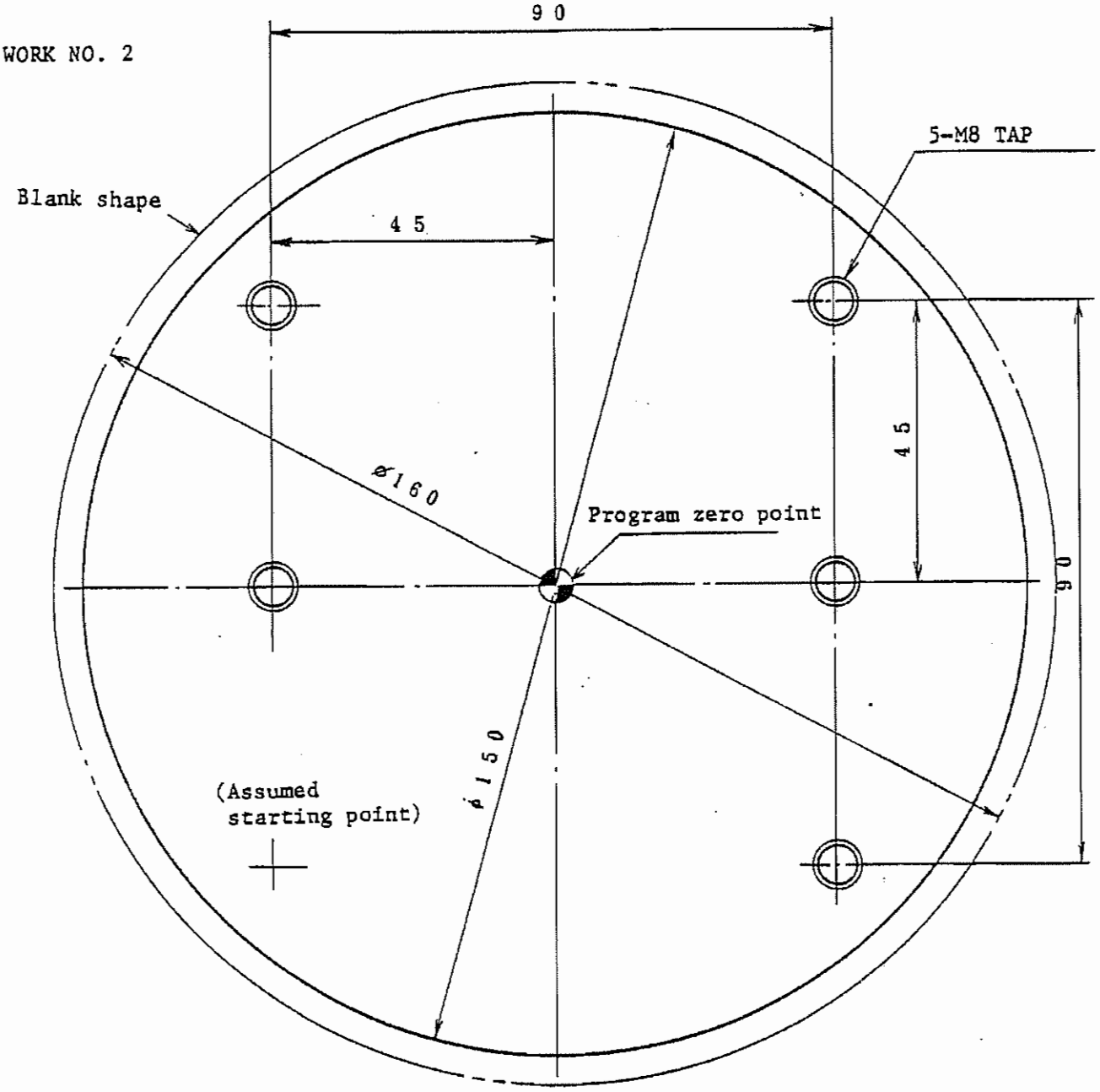
They are so designed as to be programmable if you follow the steps in the operation frame of program sheets.

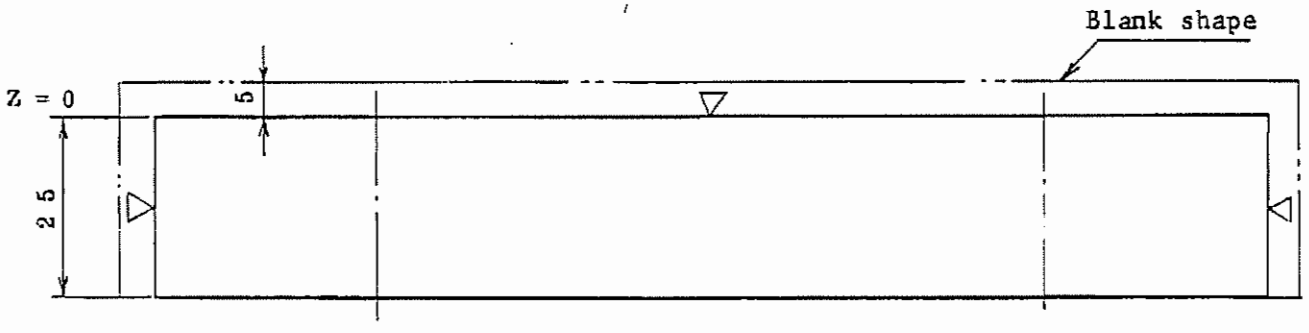
If the [INPUT] only is specified in the operation frame, the same will result also from depressing [⇒] .

- o If a wrong numeral key has been depressed (without the [INPUT] depressed yet), depress the [CLEAR] and re-enter a correct value.
- o If a wrong menu key ([ ]) has been depressed, or if a wrong numerical value has been entered (with the [INPUT] depressed after the numeral key), return the cursor by [⇐] and operate anew.

  To erase an inputted numerical value, depress [·] and [CLEAR].
- o The portion underlined in the picture is the position where a numerical value is to be inputted or a menu key is to be selected.

WORK NO. 2





MACHINING PROCEDURE

(1) MILL FRONT FACE

(2) END MILL ON THE SIDE

(3) TAP M8

WORK NO. 2

UNO	MAT	INITIAL-Z	MULTI MODE	MULTI	PITCH-X	PITCH-Y
0	CBN STL	10	③ OFF	◆	◆	◆

UNO	UNIT	X	Y	O	Z	4
1	WPC-1	-100	-200	0	-350	0

UNO	UNIT	DEPTH	SRV-Z	SRV-R	BTM	WAL	FIN-Z	FIN-R
2	F-MILL	0	5	◆	1	◆	0	◆

SNO	TOOL	NOM-Ø	NO.	APRCH-X	APRCH-Y	TYPE	ZFD	DEP-Z	WID-R	C-SP	FR	M	M
R1	F-MILL	100A		?	?	Y BI-DIR	◆	5	70	24	1.35	8	

FIG	P1X/CX	P1Y/CY	P3X/R	P3Y	CN1	CN2	CN3	CN4
① CIR	② 0	③ 0	④ 80	◆	◆	◆	◆	◆

UNO	UNIT	DEPTH	SRV-Z	SRV-R	RGH	CHMF	FIN-Z	FIN-R
⑤ 3	⑥ LIN OUT	⑦ 26	⑧ 26	⑨ 5	⑩ 1	◆	0	0

SNO	TOOL	NOM-Ø	NO.	APRCH-X	APRCH-Y	TYPE	ZFD	DEP-Z	WID-R	C-SP	FR	M	M
R1	⑫ E-MILL	⑬ 20A		⑭ ?	?	⑮ CW	⑯ G00	⑰ 26	◆	⑱ 14	0.12	⑲ 8	

FIG	P1X/CX	P1Y/CY	P3X/R	P3Y	CN1	CN2	CN3	CN4
CIR	0	0	75	◆	◆	◆	◆	◆

UNO	UNIT	NOM-	MAJOR-Ø	PITCH	TAP-DEP	CHMF	CHP
4	⑳ TAP	㉑ M8	㉒ 8	㉓ 1.25	㉔ 26	㉕ 1	㉕' 0

SNO	TOOL	NOM-Ø	NO.	HOLE-Ø	HOLE-DEP	PRE-DIA	PRE-DEP	RGH	DEPTH	C-SP	FR	M	M
1.	CTR-DR	20		10	◆	◆	◆	Ⓑ ◆	◆	20	0.2		
2.	DRILL	6.9		6.9	Ⓐ 33.25	◆	◆	PCK1	T3.45	22	0.086	8	
3.	TAP	M8		8	26	◆	◆	◆	P1.25	8	125	8	

FIG	PTN	Z	X	Y	AN1	AN2	T1	T2	F	M	N	P	Q	R
1	㉖ SQR	㉗ 0	㉘ -45	㉙ -45	㉚ 0	㉛ 90	㉜ 90	㉝ 45	㉞ 0	㉟ 2	㊱ 3	㊲ 0	㊳ 1	㊴ 0

UNO	UNIT	CONTI.	NUMBER
5	END	0	0

NO.	PROGRAM MESSAGE	OPERATION	MEANING OF OPERATION
	UNO 0 ∿ 4th line (SNO R1) are omitted. See WORK NO. 1.		
1	PATTERN OF FIGURE <MENU> ?	CIRCLE	Enter a machining shape. (circle in the example)
2	CIRCLE CENTER X ?	0, INPUT	X coordinate value of machining shape center
3	CIRCLE CENTER Y ?	0, INPUT	Y coordinate value of machining shape center
4	CIRCLE RADIUS R	8, 0, INPUT	Enter a radius. (In the example, the radius is 75. This is, however, a finished size. Therefore, enter 80, with cutting allowance of 5 mm in UNO 3 taken into account.)
5	POINT CUTTING PATTERN <MENU> ?	SHAPE END	End of a machining shape definition
6	MACHINING UNIT <MENU> ?	LINE MACH-ING	Select a machining unit.
6'	MACHINING UNIT <MENU> ?	LINE OUT	Select one of LINE/MACH-ING units. (line out machining in the example)

NO.	PROGRAM MESSAGE	OPERATION	MEANING OF OPERATION
7	DIST: WPC-Z = 0 TO FIN. SURFACE ?	[2], [6], [INPUT]	Distance between program zero point and bottom to be machined (depth) (In the example, the thickness is 25 mm. Since the side is cut off, however, take an allowance of 1 mm.)
8	Z AXIS STOCK REMOVAL ?	[2], [6], [INPUT]	Z axial cutting allowance
9	X/Y AXIS STOCK REMOVAL ?	[5], [INPUT]	Radial cutting allowance
10	SURFACE ROUGHNESS <MENU> ?	[▼ 1]	Machined face roughness.
11	FINISH ALLOWANCE-Z ? FINISH ALLOWANCE-R ?	[INPUT] [INPUT]	Determine rough and/or finish machining
12	WHICH TYPE OF TOOL <MENU> ?	[INPUT] or [ENDMILL]	Tool name (questioned in the menu though automatically determined)
13	NOMINAL DIAMETER	[2], [0], [INPUT]	Tool diameter (nominal diameter)
13'	TOOL FILE CODE <MENU> ?	[A]	Set a suffix
14	APPROACH POINT X, AUTO←<MENU> ? APPROACH POINT Y, AUTO←<MENU> ?	[AUTO SET], [INPUT] (Depress [INPUT] when the cursor comes onto APPROACH POINT-Y.)	Determine an approach point (automatically determined)

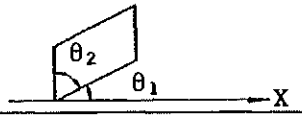
The 8th line FIG / CIRC is omitted. See ① - ⑤ .

NO.	PROGRAM MESSAGE	OPERATION	MEANING OF OPERATION
15	CUTTING DIRECTION <MENU> ?	[CW CUT]	Cutting method (In the example, go on machining in the CW direction)
16	FEEDRATE-Z, < MENU > OR < DATA > ?	[RAPID G00]	Z-axis drection feedrate selection
17	DEPTH OF CUT, AUTO→<MENU>?	[AUTO SET]	Z-axial cutting stroke per cycle (automatically determined)
18	CUTTING SPEED, AUTO→<MENU> ? FEEDRATE, AUTO→<MENU> ?	[HSS AUTO]	Circumferential speed (automatically determined) (Feedrate is also automatically determined.)
19	M CODE ?	[08 FLOOD COOLANT], [INPUT] (Depress the [INPUT] when the cursor comes to the second M code.)	
20	MACHINING UNIT <MENU> ?	[POINT MACH-ING]	
20'	MACHINING UNIT <MENU> ?	[TAP]	
21	TAP TYPE <MENU> ?	[METRIC THRD(M)]	Tap nominal diameter (Select a metric thread.)
21'	NOMINAL DIAMETER ?	[8], [INPUT]	Tap diameter

The 8th line $\frac{\text{FIG}}{\text{CIRC}}$ is omitted. See ① - ⑤.

NO.	PROGRAM MESSAGE	OPERATION	MEANING OF OPERATION
22	(ACTUAL DIA. OF TAP ?)	(INPUT)	Tap outside diameter (automatically determined, with (20) and (20') selected)
23	(PITCH ?)	(INPUT)	Tap thread pitch (automatically determined) (Enter in case of a fine thread.)
24	(TAPPING DEPTH ?)	2 , 6 , INPUT	Screw depth (work thickness 25 mm + piercing allowance 1 mm)
25	CHAMFER WIDTH ?	1 , INPUT	Chamfering at the mouth of tap
25'	CHIP VAC. CLEANER <Y: 1, N: 0> ?	0 , INPUT	
	SNO 1 ∿ 3 are omitted. With the cursor positioned at C-SP, depress HSS AUTO for any of CTR-DR, DRILL or TAP.	HSS AUTO	Note 1: (A) Tap bottom hole drill depth is automatically determined deeper. (B) Drilling cycle, high-speed deep hole drilling cycle and deep hole drilling cycle are determined automatically according to a drill hole depth.
26	POINT CUTTING PATTERN <MENU> ?	SQUARE	Enter a point-machining shape. (Select SQUARE.)
27	Z VALUE OF WORK SURFACE ?	0 , INPUT	Coordinates between program zero point with Z = 0 and face which a point is to be machined (in the example, Z = 0, because the program zero point surface is flush with the machined point face.)

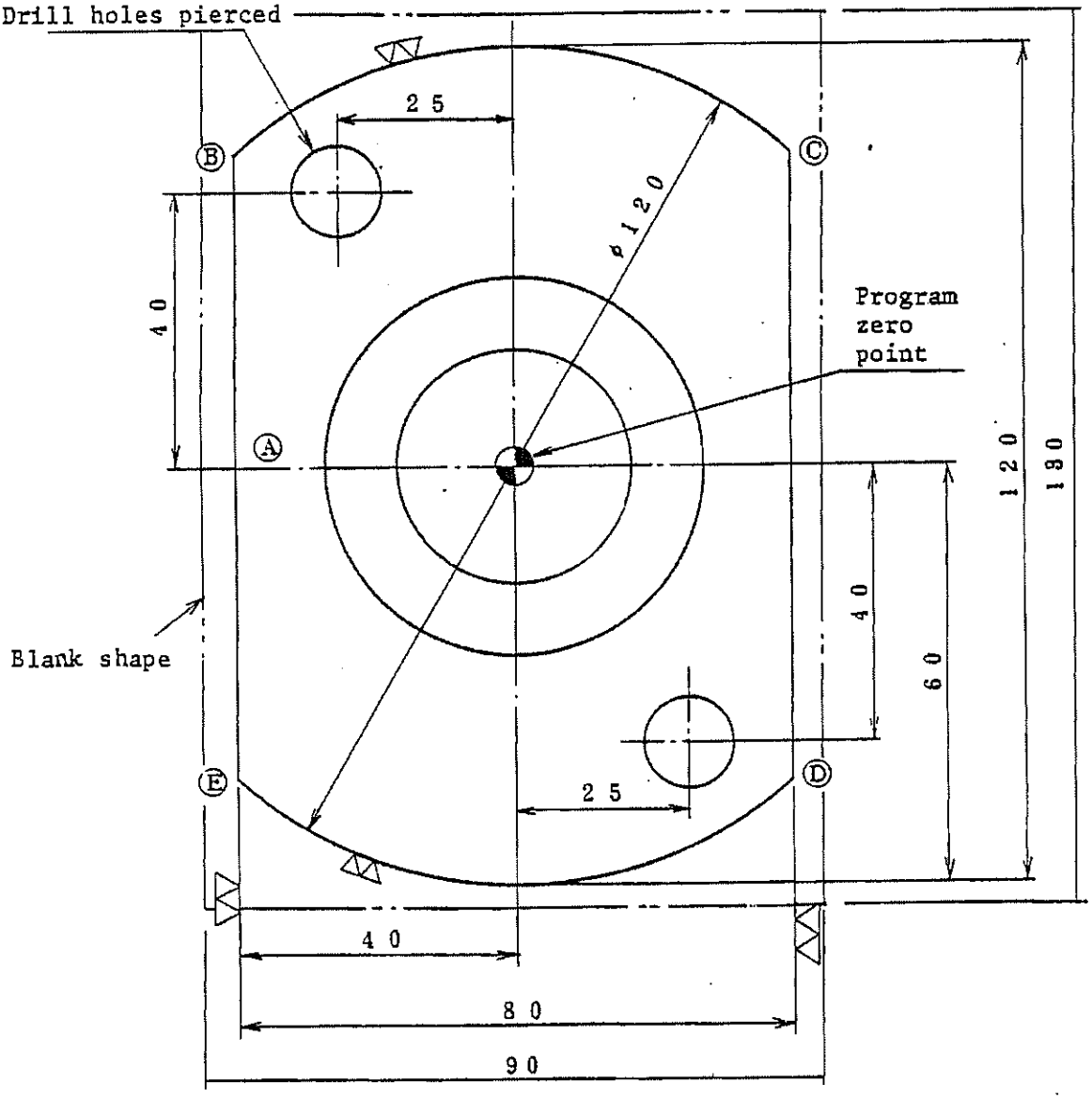
The 8th line $\frac{\text{FIG}}{\text{CIRC}}$ is omitted. See ① - ⑤.

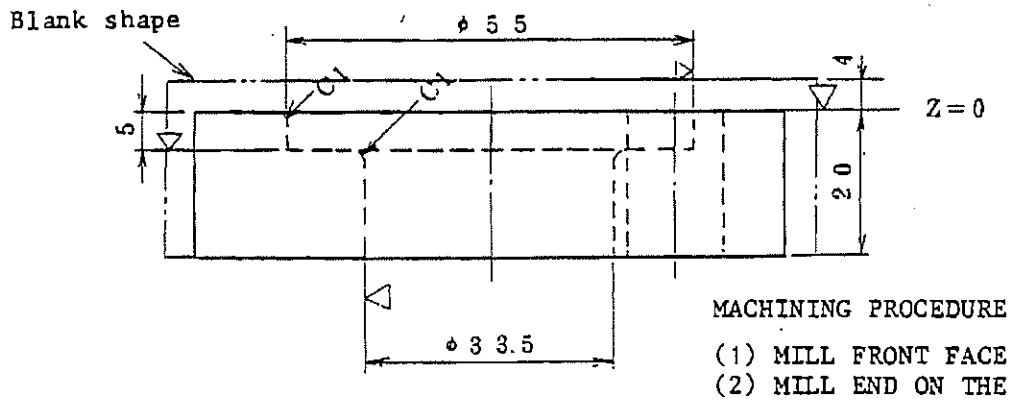
NO.	PROGRAM MESSAGE	OPERATION	MEANING OF OPERATION
28	STARTING POINT X ?	[-], [4], [5], [INPUT]	X coordinate value of square starting point (first hole)
29	STARTING POINT Y ?	[-], [4], [5], [INPUT]	Y coordinate value of square starting point (first hole)
30	ANG OF START LIN FROM X AXIS ?	[0], [INPUT]	Angle formed between first point array of square and X axis (in degrees) (In the example, θ1 = 0, because they are in parallel.)
31	ANG BETWEEN THE TWO LINES ?	[9], [0], [INPUT]	Angle formed between first and last arrays (in degrees) (In the example, θ2 = 90.) (See illustration below.)
			(Example) 
32	PITCH/LENGTH OF PATTERN AN1 ?	[9], [0], [INPUT]	Point pitch in first array (45 in the example)
33	PITCH/LENGTH OF PATTERN AN2 ?	[4], [5], [INPUT]	Point pitch in the second (last) array (45 in the example)
34	T1 & T2→<PITCH: 0, LINE LENG.:1> ?	[0], [INPUT]	0 with T1 and T2 in (31) and (32) specified in pitch 1 with T1 and T2 in (31) and (32) specified in length
35	NUMBER OF HOLES IN LINE AN1 ?	[2], [INPUT]	Number of holes on the first array (2 in the example)
36	NUMBER OF HOLES IN LINE AN2 ?	[3], [INPUT]	Number of holes on the 2nd (last) array (3 in the example)

NO.	PROGRAM MESSAGE	OPERATION	MEANING OF OPERATION
37	OMIT 4 CNR. EX SPT <Y:1, N:0> ?	[0], [INPUT]	0 with four holes at corners machined (0 in the example) 1 with no hole at the corner machined
38	OMIT SPT MACHINING <Y:1, N:0> ?	[1], [INPUT]	0 with the first hole machined (1 in the example) 1 with the first hole not machined
	POINT CUTTING PATTERN <MENU> ?	[SHAPE END]	
39	RETURN POSITION <INIT:0, R:1>?	[0], [INPUT]	Return point level selection 0:R point (parameter: BS2) return 1: Initial point return
	[UNO] is omitted. 5		

WORK NO. 3

2 Ø13 Drill holes pierced





MACHINING PROCEDURE

(1) MILL FRONT FACE
(2) MILL END ON THE SIDE
(3) DRILL Ø13 HOLES
(4) BORING Ø33.5
(5) ENDMILL Ø55

WORK NO. 3

UNO	MAT	INITIAL-Z	MULTI MODE	MULTI	PITCH-X	PITCH-Y
0	CBN STL	10	OFF	◆	◆	◆

UNO	UNIT	X	Y	Q	Z	4
1	WPC-1	-100	-200	0	-300	0

UNO	UNIT	DEPTH	SRV-Z	SRV-R	BTM	WAL	FIN-Z	FIN-R
2	F-MILL	0	4	◆	1	◆	0	◆

SNO	TOOL	NOM-Ø	NO.	APRCH-X	APRCH-Y	TYPE	ZFD	DEP-Z	WID-R	C-SP	FR	M	M
R1	F-MILL	100A		?	?	X BI-DIR	◆	4	70	24	1.35	8	

FIG	P1X-CX	P1Y/CY	P3X/R	P3Y	CN1	CN2	CN3	CN4
SQR	-45	-65	45	65	R0	R0	R0	R0

UNO	UNIT	DEPTH	SRV-Z	SRV-R	RGH	CHMF	FIN-Z	FIN-R
3	LIN-OUT	21	21	5	4	◆	0.6	0.6

SNO	TOOL	NOM-Ø	NO.	APRCH-X	APRCH-Y	TYPE	ZFD	DEP-Z	WID-R	C-SP	FR	M	M
R1	E-MILL	30A		?	?	CW	G01	20.4	◆	16	0.448	8	
F2	E-MILL	30A		?	?	CW	G01	◆	◆	18	0.229	8	

FIG	PTN	X	Y	R/O	I	J	P	CNR
②1	③LINE	④-40	⑤ 0					
2	⑥LINE	⑦-40	⑧ ?	⑨ 90			⑩ UP	
3	⑪ CW	⑫ 40	⑬ ?	⑭ 60	⑮ 0	⑯ 0	⑰ UP	
4	LINE	40	?	90			DOWN	
5	CW	-40	?	60	0	0	DOWN	
6	LINE	-40	0					

UNO	UNIT	DIA	DEPTH	CHMF
4	DRILL	13	21	1

SNO	TOOL	NOM-Ø	HOLE-Ø	HOLE-DEP	PRE-DIA	PRE-DEP	RGH	DEPTH	C-SP	FR	M	M
1	CTR-DR	20	15	◆	◆	◆	◆	◆	20	0.2	8	
2	DRILL	13	13	21	◆	◆	DRILL	T21	25	0.153	8	

FIG	PTN	Z	X	Y	AN1	AN2	T1	T2	F	M	N	P	Q	R
1	PT	0	-25	40	◆	◆	◆	◆	◆	◆	◆	0	0	C
2	PT	0	25	-40	◆	◆	◆	◆	◆	◆	◆	0	0	0

UNO	UNIT	DIA	DEPTH	CHMF	WAL
5 ⑱	BORE T1	⑲ 33.5	⑳ 20	㉑ 0	㉒ 2

SNO	TOOL	NOM-Ø	HOLE-Ø	HOLE-DEP	PRE-DIA	PRE-DEP	RGH	DEPTH	C-SP	FR	M	M
1	CTR-DR	20	10	◆	◆	◆	◆	◆	20	0.2	8	
2	DRILL	27.5	27.5	21	◆	◆	DRILL	T27.5	25	0.252	8	
3	BOR BAR	33.5	33.5	21	◆	0	2	T3	74	0.144	8	

FIG	PTN	Z	X	Y	AN1	AN2	T1	T2	F	M	N	P	Q	R
1	PT	0	0	0	◆	◆	◆	◆	◆	◆	◆	0	0	0

UNO	UNIT	DIA	DEPTH	CHMF	BTM	PRE-DIA		CHMF
6 ㉓	CIRC MIL	㉔ 55	㉕ 5	㉖ 1	㉗ 1	㉘ 33.5		㉙ 1

SNO	TOOL	NOM-Ø	HOLE-Ø	HOLE DEP	PRE-DIA	PRE-DEP	RGH	DEPTH	C-SP	FR	M	M
1 ㉚	E-MILL	㉛ 30A	㉜ 55	㉝ 5	㉞ 33.5	◆	㉟ 1	㊱ T16.8	㊲ 17	0.282	㊳ 8	
2 ㊴	CHMF	㊵ 30A	㊶ 999	㊷ 0	㊸ 55	㊹ 5	◆	㊺ C1	㊻ 27	0.3	㊼ 8	
3	CHMF	30A	55	5	33.5	999	◆	C1	27	0.3	8	

FIG	PTN	Z	X	Y	AN1	AN2	T1	T2	F	M	N	P	Q	R
1	PT	0	0	0	◆	◆	◆	◆	◆	◆	◆	0	0	0

UNO	UNIT	CONTI.	NUMBER
7	END	0	0

NO.	PROGRAM MESSAGE	OPERATION	MEANING OF OPERATION
	UNO ~ UNO (up to 6th line) 0 3 are omitted.		
	See Work Nos. 1 and 2		
1	FINISH ALLOWANCE-Z ? FINISH ALLOWANCE-R ?	INPUT INPUT	
	E-MILL is omitted below.		
2	PATTERN OF FIGURE <MENU> ?	ARBITRY	Machining shape (arbitry in the example)
3	POINT CUTTING PATTERN <MENU> ?	LINE	Select a shape. (The first, however, represents the starting point.)
4	COORDINATE X OF FIGURE ?	-, 4, 0, INPUT	X coordinate value of starting point (point Ⓐ in the figure)
5	COORDINATE Y OF FIGURE ?	0, INPUT	Y coordinate value of starting point
		↓	Subsequently, no entry is required. Therefore, depress the ↓ and shift to the lower line.
6	PATTERN OF FIGURE <MENU> ?	LINE	Shapes Ⓐ from Ⓑ (straight line in the example)

NO.	PROGRAM MESSAGE	OPERATION	MEANING OF OPERATION
7	COORDINATE X OF FIGURE ?	[-], [4], [0], [INPUT]	X coordinate value of point Ⓑ
8	COORDINATE Y OF FIGURE ?	[?]	Y coordinate value of point Ⓑ ([?] may by entered unless this value is displayed on the picture.)
9	ANGLE FROM X AXIS ?	[9], [0], [INPUT]	Angle formed between straight line Ⓐ → Ⓑ and X axis. (in degree)
		[→] [→]	Skip the cursor over I and J:
10	INTERSEC.PT LINE-CTR <MENU> ?	[UP]	Set a modifier. (The modifier should be specified if there are a certain number of crossings.)
		[↓]	Move the cursor downward.
11	PATTERN OF FIGURE <MENU> ?	CW ARC [◯]	Shape Ⓑ → Ⓒ (CW arc in the example)
12	COORDINATE X OF FIGURE ?	[4], [0], [INPUT]	X coordinate value of Ⓒ
13	COORDINATE Y OF FIGURE ?	[?]	Y coordinate value of Ⓒ
14	RADIUS R ?	[6], [0], [INPUT]	Radius of arc Ⓑ → Ⓒ

NO.	PROGRAM MESSAGE	OPERATION	MEANING OF OPERATION
15	CIRC. INTERPOLATION CENTER X ?	[0], [INPUT]	X coordinate value of arc center
16	CIRC. INTERPOLATION CENTER Y ?	[0], [INPUT]	Y coordinate value of arc center
17	INTERSEC.PT LINE/CTR<MENU>?	[UP]	Set a modifier
		↓	Move the cursor downward. (Subsequently, set Ⓓ→Ⓔ→Ⓐ in that order.)
	Likewise, proceed to [FIG 4] ∿ [FIG 6]	(Before [UNO 4] after completion of [FIG 6]) [SHAPE END]	
	[UNO 4] ∿ [UNO 2] are omitted.		
18	MACHINING UNIT <MENU> ?	[POINT MACH-ING]	Machining unit.
18'	MACHINING UNIT <MENU> ?	[BORING □]	Enter point machining unit
18''	MACHINING UNIT <MENU> ?	[BORING □]	Enter classification in boring unit
19	HOLE DIAMETER ?	[3], [3], [.], [5], [INPUT]	Boring hole diameter

NO.	PROGRAM MESSAGE	OPERATION	MEANING OF OPERATION
20	HOLE DEPTH ?	[2], [0], [INPUT]	Boring hole depth.
21	CHAMFER WIDTH ?	[0], [INPUT]	Stroke of chamfering at the mouth (without chamfering in the example)
22	WALL ROUGHNESS ▙ <MENU> ?	[▼ 2]	Roughness of boring finish face.
	CTR-DR, DRILL and BOR BAR following [SNO] 1 are omitted		
	With the cursor positioned at C-SP, [HSS AUTO] for CTR-DR and DRILL and [CARBIDE AUTO] for BOR BAR		
	LINE [FIG] 1 PTN is omitted PT		
23	MACHINING UNIT <MENU> ?	[POINT MACH-ING]	Enter a machining unit
23'	MACHINING UNIT <MENU> ?	[CIRC-MIL (0)]	POINT MACH-ING unit
24	HOLE DIAMETER ?	[5], [5], [INPUT]	Machining hole diameter
25	HOLE DEPTH ?	[5], [INPUT]	Machining hole depth

NO.	PROGRAM MESSAGE	OPERATION	MEANING OF OPERATION
26	CHAMFER WIDTH ?	[1], [INPUT]	Stroke of chamfering at the mouth.
27	BOTTOM ROUGHNESS <MENU> ?	[▼ 1]	Bottom finish face roughness.
28	PREPARED HOLE DIAMETER ?	[3],[3],[.],[5], [INPUT]	Bottom hole diamter.
29	CHAMFER WIDTH ?	[1], [INPUT]	Stroke of chamfering at the mouth of bottom hole.
30	WHICH TYPE OF TOOL <MENU> ?	[INPUT] , (or [ENDMILL])	Tool name (automatically determined)
31	NOMINAL DIAMTER ?	[3],[0], [INPUT]	Tool nominal diameter
31'	TOOL FILE CODE <MENU> ?	[A]	Suffix
32	HOLE DIAMETER ?	[INPUT]	Circle machining hole diameter (automatically determined)
33	HOLE DEPTH ?	[INPUT]	Circle machining hole depth (automatically determined)
34	PREPARED HOLE DIAMETER ?	[INPUT]	Bottom hole diameter (automatically determined)

NO.	PROGRAM MESSAGE	OPERATION	MEANING OF OPERATION
35	BOTTOM ROUGHNESS <MENU> ?	INPUT	Circle machining bottom roughness (automatically determined)
36	DEPTH OF CUT, AUTO+<MENU> ?	INPUT	Axial cutting stroke per cycle (automatically determined)
37	CUTTING SPEED, AUTO+<MENU> ? FEEDRATE, AUTO+<MENU> ?	HSS AUTO	Circumferential speed (automatically determined) (Feed is also automatically determined at a time.)
38	M CODE ?	08 FLOOD COOLANT, INPUT	Auxiliary function (M code)
39	WHICH TYPE OF TOOL <MENU> ?	INPUT (or CHAMF. CUTTER)	Tool name (automatically determined).
40	NOMINAL DIAMETER ?	3, 0, INPUT	Tool nominal diameter
40'	TOOL FILE CODE <MENU> ?	A	Suffix(without suffix in the example).
41	TOOL PERIMETER INTERFERENCE ?	INPUT	Interference diameter (interference of chamfering cutter) (not expecially entered)
42	DIST: FROM Z=0 TO CUT SURFACE ?	INPUT	Chamfering depth(automatically determined)

NO.	PROGRAM MESSAGE	OPERATION	MEATING OF OPERATION
43	PREPARED HOLE DIAMETER ?	INPUT	Hole diameter at chamfered portion (automatically determined)
44	PREPARED HOLE DEPTH ?	INPUT	Machining depth at chamfered portion (automatically determined) (ø55 in the example)
45	CHAMFER WIDTH ?	INPUT	Chamfering stroke of bottom hole (automatically determined)
46	CUTTING SPEED, AUTO→<MENU> ? (FEEDRATE, AUTO→<MENU> ?)	HSS AUTO	Circumferential speed (automatically determined)
47	M CODE ?	FLOOD COOLANT , INPUT	M code(auxiliary function)
	Subsequently up to 7 UNO are omitted.		

5. MISCELLANEOUS FUNCTION (M FUNCTION)

◎ Option

M CODE	FORMAT	HQC	H-12	H-15	H-22	H25Q	H-15J	H-25SP	V-7.5	V-10TS V-10N	V-15N	V-20	VQC		
00	Program stop	○	○	○	○	○	○	○	○	○	○	○	○		
01	Optional program stop	○	○	○	○	○	○	○	○	○	○	○	○		
02	End of program (EIA/ISO)	○	○	○	○	○	○	○							
03	Spindle CW start	○	○	○	○	○	○	○	○	○	○	○	○		
04	Spindle CCW start	○	○	○	○	○	○	○	○	○	○	○	○		
05	Spindle stop	○	○	○	○	○	○	○	○	○	○	○	○		
06	Tool change (EIA/ISO)		○	○	○	○	○	○	○	○	○	○			
07	Oil mist ON	○	○	○	○	○	○	○	○	○	○	○	○		
08	Flood coolant ON	○	○	○	○	○	○	○	○	○	○	○	○		
09	Oil mist/Flood coolant/Air blast/Chip removal/Oil hole adapter/Tapping coolant OFF	○	○	○	○	○	○	○	○	○	○	○	○		
10	Tool clamp	◎	◎	◎	◎	○	◎	◎							
11	Tool unclamp	◎	◎	◎	◎	○	◎	◎							
12															
13															
14															
15	Magazine cover CLOSE	○											○		
16	Magazine cover OPEN	○											○		
17															
18															
19	Spindle oriented stop	○	○	○	○	○	○	○	○	○	○	○	○		
20	Special head clamp					○		○							
21	Special head unclamp					○		○							
22															

◎ Option

M CODE	FORMAT	HQC	H-12	H-15	H-22	H25Q	H-15J	H-25SP	V-7.5	V-10TS V-10N	V-15N	V-20	VQC		
23	Error detect ON	O	O	O	O	O	O	O	O	O	O	O	O		
24	Error detect OFF	O	O	O	O	O	O	O	O	O	O	O	O		
25	Drum shifter retract					O		◎							
26															
27	Drum load					O		◎							
28	Drum unload					O		◎							
29															
30	Reset and rewind EIA/ISO)	O	O	O	O	O	O		O	O	O	O	O		
31	Cancel interlock					O									
32	M31 cancel					O									
33															
34															
35	Tool breakage search	O	O	O	O	O	O	O	O	O	O	O	O		
36	Spindle low gear					O									
37	Spindle range "Low" Spindle range "Low middle"(Only for H-25Q)			O	O	O	O								
38	Spindle range "Middle/Low" Spindle range "High middle"	O/	O/	O/	O/	O	O/	O/	O/	O/	O/	O/	O/		
39	Spindle range "High"	O	O	O	O	O	O	O	O	O	O	O	O		
40	Spindle neutral			O	O			O							
41															
42	Index table CCW						O								
43	Index table command 3			◎	◎	◎	◎		◎	◎	◎	◎	◎		
44	Index table command 1			◎	◎	◎	◎		◎	◎	◎	◎	◎		
45	Index table command 2			◎	◎	◎	◎		◎	◎	◎	◎	◎		

◎ Option

M CODE	FORMAT	HQC	H-12	H-15	H-22	H25Q	H-15J	H-25SP	V-7.5	V-10TS V-10N	V-15V	V-20	VQC		
46															
47															
48	Cancel M49 (EIA/ISO)						○								
49	Override cancel (EIA/ISO)						○								
50	Air blast start	○	○	○	○	○	○	○	○	○	○	○	○		
51	Coolant through spindle ON	◎	◎	◎	◎	◎	◎	◎	◎	◎	◎	◎	◎		
52	Tap coolant	◎	◎	◎	◎	◎	◎	◎	◎	◎	◎	◎	◎		
53	Air hole cool/adaptor	◎	◎	◎	◎	◎	◎	◎	◎	◎	◎	◎	◎		
54	Chip air blow/Workpiece cleaning	○	○	○	○	○		○							
55															
56															
57															
58	Tool life monitoring function	○	○	○	○	○	○	○	○	○	○	○	○		
59	Program end ready					○	○								
60		○													
61	Pallet load	○	○	○	○	○	○	○							
62	Pallet unload	○	○	○	○	○	○	○							
63															
64	Pallet door close	○						○	◎	◎	◎	◎	◎		
65	Pallet door open	○						○	◎	◎	◎	◎	◎		
66	Pallet fork extend		○	○	○		○								
67															
68	Pallet clamp	○	○	○	○	○	○	○	○	○	○	○	◎		
69	Pallet unclamp	○	○	○	○	○	○	○	○	○	○	○	◎		

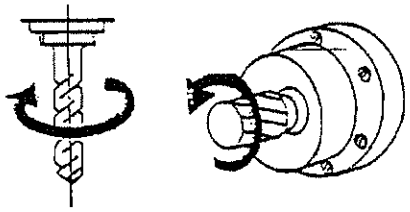
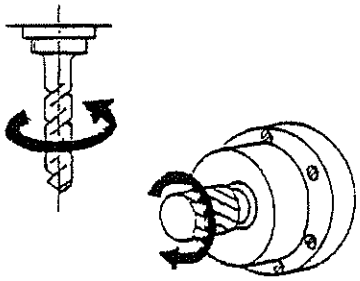
◎ Option

M CODE	FORMAT	HQC	H-12	H-15	H-22	H25Q	H-15J	H-25SP	V-7.5	V-10TS V-10N	V-15N	V-20	VQC		
70			◎	◎	◎										
71	Pallet No.1 selection	◎	◎	◎	◎	○	◎		◎	◎	◎	◎	◎		
72	Pallet No.2 selection	◎	◎	◎	◎	○	◎		◎	◎	◎	◎	◎		
73	Pallet No.3 selection	◎	◎	◎	◎		◎			◎	◎	◎	◎		
74	Pallet No.4 selection	◎	◎	◎	◎		◎			◎	◎	◎	◎		
75	Pallet No.5 selection	◎	◎	◎	◎		◎						◎		
76	Pallet No.6 selection	◎	◎	◎	◎		◎								
77	Pallet No.7 selection		◎	◎	◎		◎								
78	Pallet No.8 selection		◎	◎	◎		◎								
79	Pallet No.9 selection		◎	◎	◎		◎								
80	Pallet No.10 selection						◎								
81	Pallet No.11 selection						◎								
82	Pallet No.12 selection						◎								
83	Pallet No.13 selection						◎								
84	Pallet No.14 selection						◎								
85	Pallet No.15 selection						◎								
86	Pallet No.16 selection						◎								
87	Special head tool clamp														
88	Special head tool clamp					○									
89	Special head tool unclamp					○									
90	Mirror image cancel	○	○	○	○	○	○	○	○	○	○	○	○		
91	Mirror image X ON	○	○	○	○	○	○	○	○	○	○	○	○		
92	Mirror image Y ON	○	○	○	○	○	○	○	○	○	○	○	○		
93															
94															

◎ Option

M CODE	FORMAT	HQC	H-12	H-15	H-22	H25Q	H-15J	H-25SP	V-7.5	V-10TS V-10N	V-15N	V-20	VQC		
95															
96															
97															
98	Call sub-program (EIA/ISO)	O	O	O	O	O	O	O	O	O	O	O	O		
99	End of sub-program (EIA/ISO)	O	O	O	O	O	O	O	O	O	O	O	O		
100	For index externally attached												◎		
101	For index externally attached												◎		

List of M-code

CODE	FUNCTION	DESCRIPTION
M00	Program stop	After executing the block for which M00 has been commanded, the code stops the automatic operation. All modal information will be stored.
M01	Optional program stop	After executing the block for which M01 has been commanded, the code stops the automatic operation, provided, however, that the optional stop switch on the control panel is ON. Like M00, all modal information is stored.
M02	End of program (EIA/ISO)	This code causes the EIA/ISO program to stop running while resetting the NC unit.
M03	Spindle CW start	 V-Type H-Type This command causes the spindle to turn in the direction illustrated.
M04	Spindle CCW start	 V-Type H-Type This command causes the spindle to turn in the direction illustrated.
M05	Spindle stop	This command causes the spindle to stop turning.
M06	Tool change (EIA/ISO)	This code causes the EIA/ISO program to return to a specified magazine the tool attached to the spindle after replaceing it with the tool loaded onto the auxiliary by Txx beforehand.
M07	Oil mist ON	This command causes a cutting fluid mist to be discharged.

CODE	FUNCTION	DESCRIPTION
M08	Coolant Flood ON	This command causes liquid cutting fluid to be discharged.
M09	Oil mist/Flood coolant/Air blast/ Chip removal/Oil hole adapter/Tapping coolant OFF	Turning OFF oil mist, flood coolant, work air blast, oil hole adapter, tapping coolant, chip removal operations by commands M07, M08, M50 and M51, M52, M53 and M54.
M10	Tool clamp	This command causes the spindle to clamp a tool.
M11	Tool unclamp	This command causes the spindle to unclamp a tool.
M12		
M13		
M14		
M15	Magazine cover CLOSE	The magazine cover closes.
M16	Magazine cover OPEN	The magazine cover opens. (Tool change possible condition)
M17		
M18		
M19	Spindle oriented stop	This command causes the spindle to stop in a place. This command is used when it is necessary during the ATC operation to cause the position of the tool key way to coincide with that of the key at the end of the spindle. Do not give a spindle orient command with the gear in neutral position. Be sure to do it with the gear in low, middle or high position.
M20	Special head clamp	It is used in the special head change cycle, not used normally.
M21	Special head unclamp	M20 clamps special head. M21 unclamps special head.

CODE	FUNCTION	DESCRIPTION
M22		
M23	Error detect ON	The corner will be machined sharp by the command of M23 Tool stops in a moment. This input of M24 or turning of the power.
M24	Error detect OFF	Cancel M23. M24 has already been selected when power is turned on, the corner will be machined in a round manner by this command. The spindle does not stop at the corner.
M25	Drum shifter retract	Give this command after confirming that the shift pin has retracted, and the drum shifter will retract. This command is used only when the shifter has stopped operation due to troubles during the drum change cycle. This is not used normally.
M26		
M27 M28	Drum load Drum unload	This command is used in the drum change cycle, not used usually. DRUM LOAD command causes the drum to be transferrered from the stock and mounted on the machine and DRUM UNLOAD command causes the drum to be returned to the stocker from the machine.
M29		

CODE	FUNCTION	DESCRIPTION
M30	Reset and rewind (EIA/ISO)	This code causes the EIA/ISO program to stop running while resetting the NC unit. At the same time, it causes the magnetic tape to be rewound in the TAPE OPERATION mode.
M31	Cancel interlock	It is used in a drum changer cycle with a user macro. This command cancels ordinary +OVERTRAVEL signal of Z axis. It is not used usually.
M32	M31 cancel	It cancels M31.
M33		
M34		
M35	Tool breakage search command	To give the tool breakage search command to the NC unit. (unusable for the EIA/ISO programs)
M36	Spindle range "Low"	This command causes the spindle to turn in a low speed range.
M37	Spindle range "Low" Spindle range "Low middle" (only for H-25Q)	This command causes the spindle to turn in a low speed range. This command causes the spindle to turn in a low middle speed range.
M38	Spindle range "Middle/Low" Spindle range "High middle" (only for H-25Q)	This command causes the spindle to turn in a middle and low speed range. This command causes the spindle to turn in a high middle speed range.
M39	Spindle range "High"	This command causes the spindle to turn in a high speed range.
M40	Spindle neutral	Shifts the spindle gear to the neutral position.
M41		
M42		

CODE	FUNCTION	DESCRIPTION
M43	Index table command 3 (option)	To give a command to the externally mounted index table by this command. (optional function)
M44	Index table command 1 (option)	To give a command to the externally mounted index table by this command. (optional function)
M45	Index table command 2 (option)	To give a command to the externally mounted index table by this command. (optional function)
M46		
M47		
M48	Cancel M49 (EIA/ISO)	This code causes the EIA/ISO program to invalidity the OVERRIDE CANCEL function specified by the M49.
M49	Override cancel (EIA/ISO)	This code cancels an OVERRIDE in the EIA/ISO program.
M50	Air blast start	This command causes AIR BLAST to operate and air to be discharged.
M51	Coolant through spindle/oil hole coolant start (option)	Commanding the M51 will permit liquid coolant to be discharged cut of the spindle.
M52	Tap coolant (option)	This command causes the cutting fluid to be discharged.
M53	Air hole cool/ adaptor ON	To command suction when a chip sucking tool is used.
M54	Chip air blow	By this command, the automatic tool shank cleaning device is switched on. It can be cancelled by M09.
	Workpiece cleaning	The workpiece claning unit moves downward to apply coolant and air to the workpiece and then moves upward.

CODE	FUNCTION	DESCRIPTION
M55		
M56		
M57		
M58	Spare tool check	With the M58 inputted, the machine will make a single block stop when a life expired tool is specified in the ptorgram. (optional function)
M59	Program end ready	To give the program end ready command to the externally CPU. (option) This command is used in FMS, not used usually.
M60		
M61	Pallet load	Commanding M61 will make the pallet in the pallet stand transported to on the table. Pallet clamping and unclamping are done automatically.
M62	Pallet unload	Commanding M62 will make the pallet on the table transported to on the pallet stand. Pallet clamping and unclamping are done automatically.
M63		
M64	Pallet door close	Closes the pallet door by this command.
M65	Pallet door open	Opens the pallet door by this command.
M66	Pallet fork extend	Extends the pallet fork by this command.
M67		
M68	Pallet clamp	Commanding M68 will make the pallet be clamped.
M69	Pallet unclamp	Commanding M69 will make the pallet be unclamped.
M70		

CODE	FUNCTION	DESCRIPTION
M71	Pallet No.1 selection (option)	These commands select the pallet number in the case of the multiple pallet changer.
M72	Pallet No.2 selection (option)	
M73	Pallet No.3 selection (option)	
M74	Pallet No.4 selection (option)	
M75	Pallet No.5 selection (option)	
M76	Pallet No.6 selection (option)	
M77	Pallet No.7 selection (option)	
M78	Pallet No.8 selection (option)	
M79	Pallet No.9 selection (option)	
M80	Pallet No.10 selection (option)	
M81	Pallet No.11 selection (option)	
M82	Pallet No.12 selection (option)	
M83	Pallet No.13 selection (option)	

CODE	FUNCTION	DESCRIPTION
M84	Pallet No.14 selection (option)	These commands select the pallet number in the case of the multiple pallet changer.
M85	Pallet No.15 selection (option)	
M86	Pallet No.16 selection (option)	
M87	Special head tool clamp	It is used in the AUX.HEAD TOOL change cycle, not used usually.
M88	Special head tool clamp	It causes the special head to clamp a tool.
M89	Special head tool unclamp	It causes the special head to unclamp a tool.
M90	Mirror image cancel	Cancels the mirror image
M91	Mirror image FRM-X ON	Causes the mirror image symmetrical with respect to the WPC-X to function for all the units including the point machining mode, line machining mode, face machining mode and manual program mode units.
M92	Mirror image FRM-Y ON	Causes the mirror image symmetrical with respect to the WPC-Y to function for all the units including the point machining mode, face machining mode and manual program mode units.
M93		
M94		
M95		
M96		
M97		

CODE	FUNCTION	DESCRIPTION
M98	Sub-program call-out (EIA/ISO)	This code causes the EIA/ISO program to call a sub-program.
M99	End of sub-program	This code causes the EIA/ISO program to return to the main program at the end of a sub-program.
M100	For index externally attached	Cannot be entered together with M44 or M45 simultaneously.
M101	For inde externally attached	Cannot be entered together with M44 or M45 simultaneously.

Preparing an CAM M2 + ROTARY TABLE Program (OPTION)

The use of a rotary table attached to MAZATROL CAM M-2 is discribed.

For a rotary table, command code [4] is used. To input the code, use the menu or input data in the position indicated by the cursor.

A program will run sequentially.

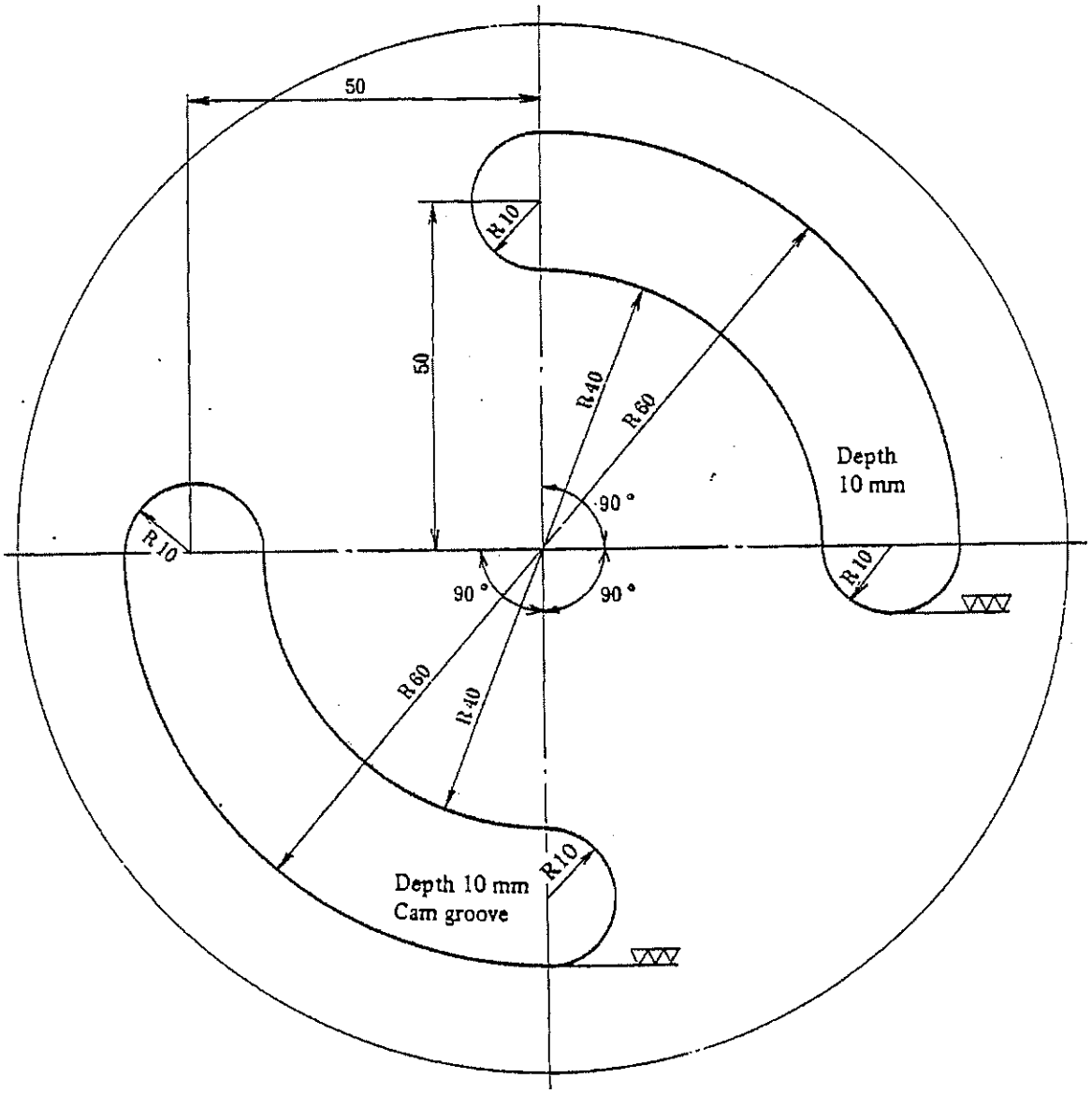
UNO	MAT	INITIAL-Z	MULTI MODE	MULTI PITCH-X	PITCH-Y
0	FCD	50.			

will appear in No. 1 block. In this case, the initial point input must be high enough so the rotary table will not interfere with any jig or workpiece even while turning.

UNO	UNIT	X	Y	Θ	Z	4
1	WPC- 0	-356.	-254.	0.	-382.	0.

When inputting WPC, X, Y and Z should be inputted by specifying one point on the workpiece to be machined. In the illustration, 0 is inputted on the 4th axis. This means that coordinates X, Y and Z are determined, with the 4th axis positioned at 0°. Θ, moreover, is independent of the 4th axis move, and should be used to input an inclination of the workpiece on the X-Y plane.

An Example of Drawing



FC

To input a work program on the 4th axis, use the manual program mode unit.

UNO	UNIT		TOOL	NOM-ϕ	NO.						
2	MANU	PRO	E-MILL	20.							
SNO	G1	G2	DATA-1	DATA-2	DATA-3	DATA-4	DATA-5	DATA-6	S	M/B	
1	90	94								M	38
2	0	Z	10.	X -50.					1000M		3
3	1	Z	-10.	F 50.							
4		4	90.	F 80.							
5	0	Z	10.								
6		4	180.								
7	1	Z	-10.	F 50.							
8		4	270.	F 80.							
9	0	Z	50.								

To machine a cam groove using the 4th axis program.

Input an angle of 90° in No. 4 block and feed F 80 as the 4th axis command.

After moying the tool upward in No. 5 block, the 4th axis returns to the zero point subject to the command in No. 6 block.

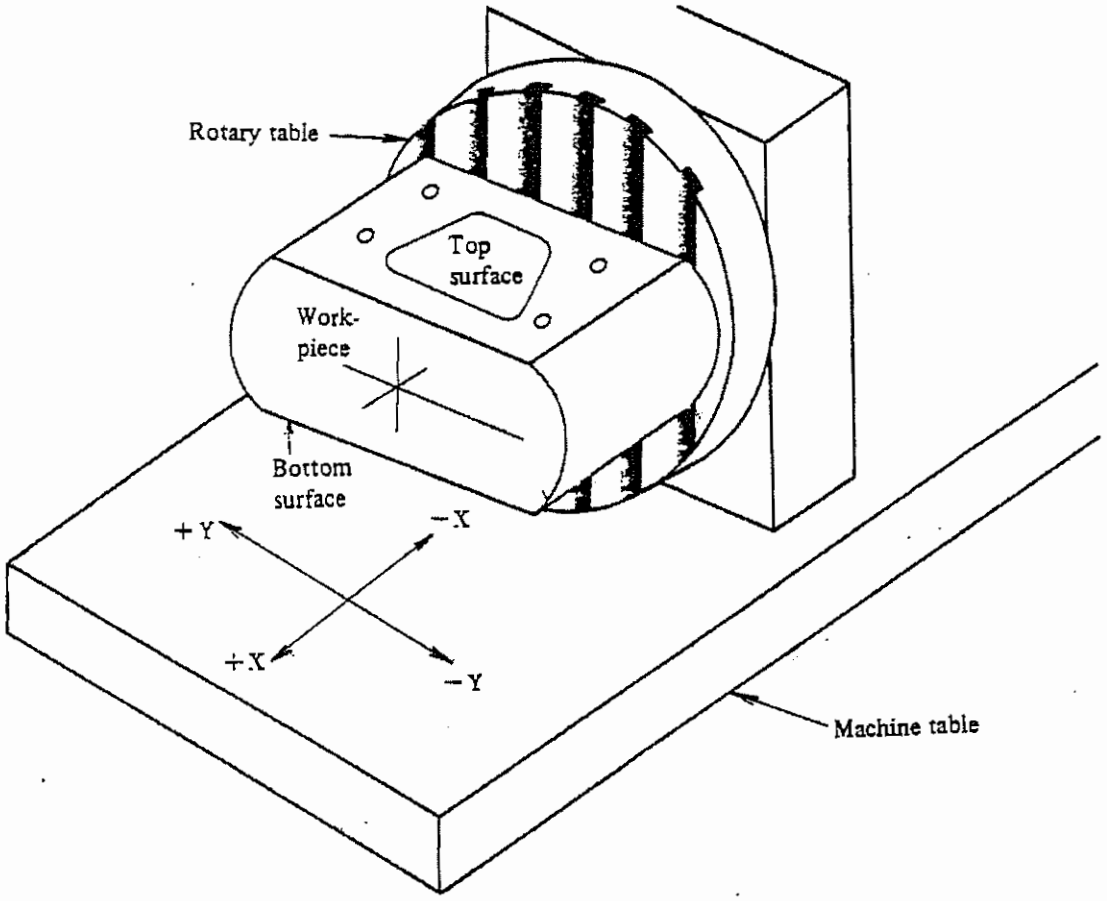
After this block, add an extra unit or complete the program in use.

NOTE: Feed F cannot be set on the 4th axis independently of the other three axis (X, Y and Z). A worm screw, cam, etc. move on the 4th axis and on the X- or Y-axis only at a constant F feed.

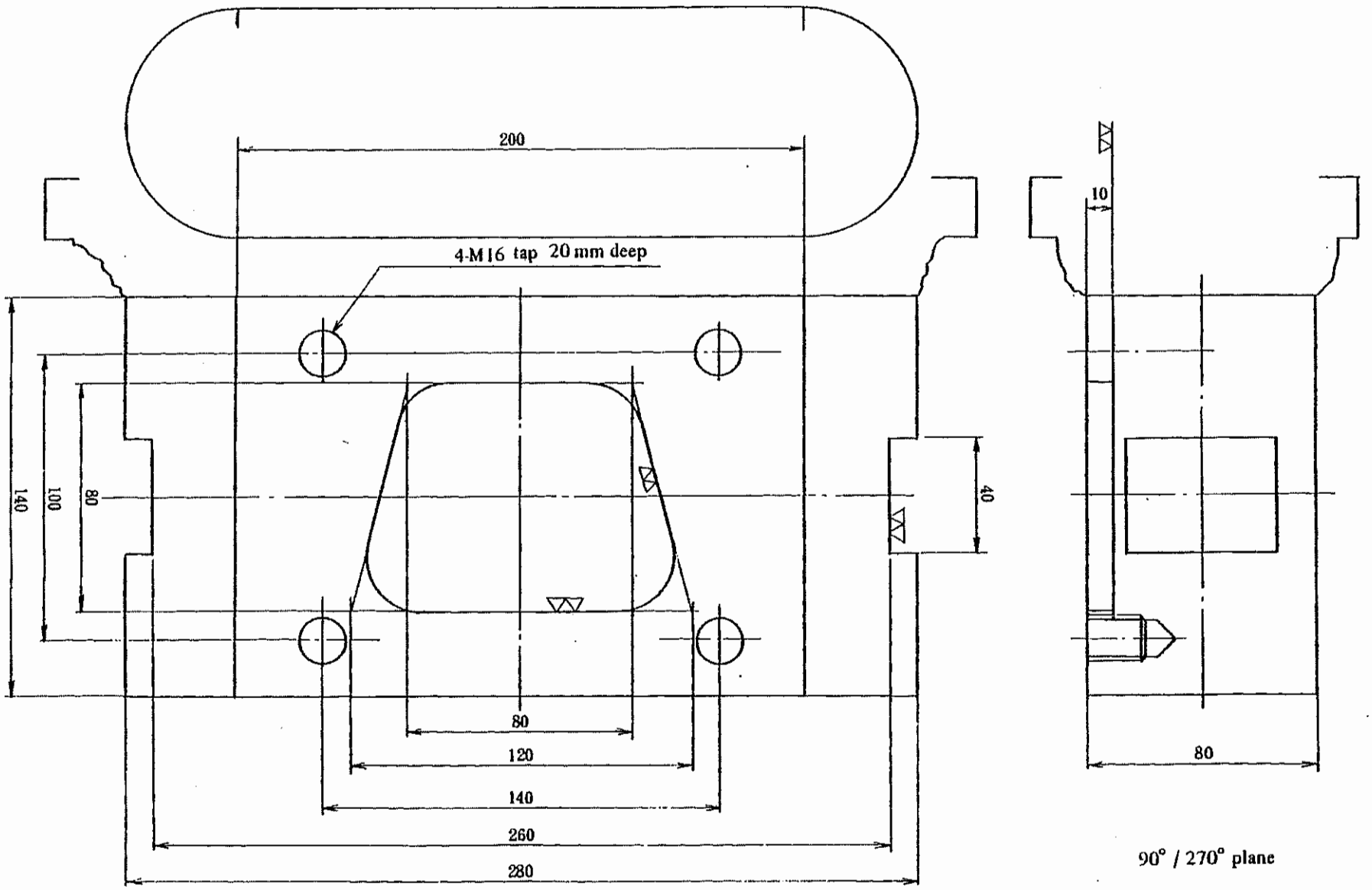
Therefore, only those workpieces which have a constant pitch can be machined.

The 4th axis command cannot be given in an automatic programming unit, only in a manual program mode unit.

Method of Programming with 4th Axis Rotary Table Used as an Indexing Table.



The rotary table can also be used as an index once the table and workpiece have been installed as illustrated above. In other words, a workpiece can be drilled and milled on both top and bottom surfaces.
The drawing of a workpiece is given below.



Drawing Example 2.

Input a common unit in unit number(UNO) 0. Input a higher value in initial point Z(INITIAL-Z) so that the rotary table will not interfere when turning.

UNO	MAT	INITIAL-Z	MULTI MODE	MULTI	PITCH-X	PITCH-Y
0	FCD	150.				

Then, input the basic coordinate system. This is an example in which inputting is started with the 4th axis positioned at 0°.

UNO	UNIT	X	Y	Θ	Z	4
1	WPC - 0	-412.	-250.	0.	-350.	0.

After that, start the work program. Machining begins with milling the plane at 0°. In milling, the line from (-100, 70) to (100, 70) in the drawing can not be done directly. This portion must be machined on the right of the line with the end mill. A little deformed shape must be inputted in milling, accordingly.

UNO	UNIT	DEPTH	SRV-Z	SRV-R	BTM	WAL	FIN-Z	FIN-R
2	F-MILL	0.	2.	↓	3	↓	0.	↓

SNO	TOOL	NOM-φ	NO.	APRCH-X	APRCH-Y	TYPE	ZFD	DEP-Z	WID-R	C-SP	FR
R 1	F-MILL	100. A		-160.	-60.	X BI-DI	↓	2.	70	85	1.314

FIG	PTN	P1X/CX	P1Y/CY	P3X/R	P3Y	CN1	CN2	CN3	CN4
1	SQR	-100.	-80.	100.	50.				

After completion of this machining, turn the workpiece. To turn it, the manual program mode unit is used.

UNO	UNIT	TOOL	NOM-φ , NO.
3	MANU PRO	F-MILL	100. A

SNO	G1	G2	DATA 1	DATA 2	DATA 3	DATA 4	DATA 5	DATA 6	S	M/B
1	00	90	A -180.							

UNO	UNIT	X	Y	Θ	Z	4
4	WPC - 0	-412.	-250.	0.	-350.	180.

The same milling is performed again. After completion of milling, no further milling is required. Keeping the 180° plane, as it is, proceed to the next machining process. Change tools. Use the end mill and proceed to machining on the right of the line.

UNO	UNIT	DEPTH	SRV-Z	SRV-R	BTM	WAL	FIN-Z	FIN-R
5	F-MILL	0.	2.	↓	3	↓	0.	↓

SNO	TOOL	NOM-Ø	NO.	APRCH-X	APRCH-Y	TYPE	ZFD	DEP-Z	WID-R	C-SP	FR
R 1	F-MILL	100.	A	-160.	-60.	X BI-BIR	↓	2.	70	85	1.314

FIG	PTN	P1X/CX	P1Y/CY	P3X/R	P3Y	CN1	CN2	CN3	CN4
1	SQR	-100.	-80.	100.	50.				

UNO	UNIT	DEPTH	SRV-Z	SRV-R	RGH	CHMF	FIN-Z	FIN-R
6	LINE RGT	0.	2.	10.	3	↓	0.	0.

SNO	TOOL	NOM-Ø	NO.	APRCH-X	APRCH-Y	TYPE	ZFD	DEP-Z	WID-R	C-SP	FR
R 1	E-MILL	20.	A	-111.	60.	G01	2.		↓	15	0.138

FIG	PTN	X	Y	R/Θ	I	J	P	CNR
1	LINE	-100.	70.					
2	LINE	100.	70.					

On the 180° plane, there is another end mill machining process which is the pocket machining which makes a concavity at the center. Use the same end mill and follow the program.

UNO	UNIT	DEPTH	SRV-Z	SRV-R	BTM	WAL	FIN-Z	FIN-R
7	POCKET	10.	10.	↓	3	3	0.	0.

SNO	TOOL	NOM-ϕ	NO.	APRCH-X	APRCH-Y	TYPE	ZFD	DEP-Z	WID-R	C-SP	FR
R 1	E-MILL	20.	A	33.116	22.183	CW	G01	10.	12.	11	0.063

FIG	PTN	X	Y	R/Θ	I	J	P		CNR
1	LINE	40.	40.					R	20.
2	LINE	60.	-40.					R	20.
3	LINE	-60.	-40.					R	20.
4	LINE	-40.	40.					R	20.

This concludes the process of entirely milling a 180° plane. Now, turn the rotary table. According to the drawing, however, portions to be machined using the end mill are on 0°, 90°, 180° and 270° planes. Then, select the 270° plane which is nearest to the 180° plane. As a matter of course, the value of Z will change. (This change is to be measured.)

UNO	UNIT	TOOL	NOM-ϕ	NO.
8	MANU PRO	E-MILL	20.	A

SNO	G1	G2	DATA 1	DATA 2	DATA 3	DATA 4	DATA 5	DATA 6	S	M/B
1	00	90	A-270.							

UNO	UNIT	X	Y	Θ	Z	4
9	WPC- 0	-412.	-250.	0.	-250.	270.

The semi-cylindrical shape existing on the 270° plane may be formed by machining a square shape with the end mill in the linear- or plane-machining pocket mode. In this example, the semi-cylindical shape is inputted in the pocket. Of various shapes, the square requires a tool diameter compensation. It is necessary, therefore, to input a slightly larger value.

UNO	UNIT	DEPTH	SRV-Z	SRV-R	BTM	WAL	FIN-Z	FIN-R		
10	POCKET	10.	10.	↓	3	3	0.	0.		
SNO	TOOL	NOM-Ø NO.	APRCH-X	APRCH-Y	TYPE	ZFD	DEP-Z	WID-R	C-SF	FR
R 1	E-MILL	20. A	0.	-9.	CW	G01	10.	12.	11	0.063
FIG	PTN	P1X/CX	P1Y/CY	P3X/R	P3Y	CN1		CN2	CN3	CN4
1	SQR	-40.	-20.	40.	20.					

This concludes machining the 270° plane. Next, using the same end mill, move to the 0° plane. On this plane, linear machining and pocket milling must be performed to follow up the previous milling. Input the data involved.

UNO	UNIT	TOOL	NOM-ϕ	NO.						
11	MANU PRO	E-MILL	20. A							
SNO	G1 G2	DATA 1	DATA 2	DATA 3	DATA 4	DATA 5	DATA 6	S	M/B	
1	90 00	A 0.								
UNO	UNIT	X	Y	Θ	Z	4				
12	WPC- 0	-412.	-250.	0.	-350.	0.				

UNO	UNIT	DEPTH	SRV-Z	SRV-R	RGH	CHMF	DEP-Z	WID-R		
13	LINE RGT	0.	2.	10.	3	↓	0.	0.		
SNO	TOOL	NOM-Ø NO.	APRCH-X	APRCH-Y	TYPE	ZFD	DEP-Z	WID-R	C-SP	FR
R 1	E-MILL	20. A	-111.	60.	↓	G01	2.	↓	15	0.138
FIG	PTN	X	Y	R/Θ	I	J	P			CNR
1	LINE	-100.	70.							
2	LINE	100.	70.							

UNO	UNIT	DEPTH	SRV-Z	SRV-R	BTM	CHMF	DEP-Z	WID-R		
14	POCKET	10.	10.	↓	3	3	0.	0.		
SNO	TOOL	NOM-ϕ NO.	APRCH-X	APRCH-Y	TYPE	ZFD	DEP-Z	WID-R	C-SP	FR
R 1	E-MILL	20. A	33.116	22.183	CW	G01	10.	12.	11	0.063
FIG	PTN	X	Y	R/Θ	I	J	P			CNR
1	LINE	40.	40.						R	20.
2	LINE	60.	-40.						R	20.
3	LINE	-60.	-40.						R	20.
4	LINE	-40.	40.						R	20.

Then the 0° plane is completely milled, only the 90° plane remains. Continue turning the workpiece.

UNO	UNIT	TOOL	NOM-ϕ	NO.					
15	MANU PRO	E-MILL	20. A						
SNO	G1 G2	DATA 1	DATA 2	DATA 3	DATA 4	DATA 5	DATA 6	S	M/B
:	00 90	A90.							
UNO	UNIT	X	Y	Θ	Z	4			
16	WPC- 0	-412.	-250.	0.	-250.	90.			

UNO	UNIT	DEPTH	SRV-Z	SRV-R	BTM	WAL	FIN-Z	FIN-R
17	POCKET	10.	10.	↓	3	3	0.	0.

SNO	TOOL	NOM-φ	NO.	APRCH-X	APRCH-Y	TYPE	ZFD	DEP-Z	WID-R	C-SP	FR
R 1	E-MILL	20.	A	0.	-9.	CW	G01	10.	12.	11	0.063

FIG	PTN	P1X/CX	P1Y/CY	P3X/R	P3Y	CN1	CN2	CN3	CN4
1	SQR	-40.	-20.	40.	20.				

Now, all milling operations have been completed. 0° and 180° planes remain untapped, so, index the 180° plane and perform the tapping operation.

UNO	UNIT	TOOL	NOM-φ	NO.
17	MANU PRO	CTR-DR	20.	A

SNO	G1	G2	DATA 1	DATA 2	DATA 3	DATA 4	DATA 5	DATA 6	S	M/B
1	90	00	A 180.							

UNO	UNIT	X	Y	Θ	Z	4
18	WPC- 0	-412.	-250.	0.	-350.	160.

UNO	UNIT	NOM.	MAJOR-φ	PITCH	TAP-DEP	CHMF	CHP
18	TAP	M 16.	16.	2.	20.	0.8	0

SNO	TOOL	NOM-φ	NO.	HOLE-φ	HOLE-DEP	PRE-DIA	PRE-DEP	RGH	DEPTH	C-SP	FR	M	M
1	CTR-DR	20.	A	16.	↓	↓	↓	90	↓	14	0.2		
2	DRILL	14.2	A	14.2	31.	↓	↓	DRILL	T31.	18	0.179		
3	TAP	M 16.	A	16.	20.	↓	↓	FIX	P2.	6	2.		

FIG	PTN	Z	X	Y	AN1	AN2	T1	T2	F	M	N	P	Q	R
1	SQR	0.	-70.	-50.	0.	90.	140.	100.	0	2	2	0	0	0

Next, tap the 0° plane. Now all the processes have been completed.

UNO	UNIT	TOOL	NOM-φ	NO.
19	MANU PRO	CTR-DR	20.	A

SNO	G1	G2	DATA 1	DATA 2	DATA 3	DATA 4	DATA 5	DATA 6	M	S/B
1	90	00	A 0.							

UNO	UNIT	X	Y	Θ	Z	4
20	WPC- 0	-412.	-250.	0.	-350.	0.

UNO	UNIT	NOM.	MAJOR-φ	PITCH	TAP-DEP	CHMF	CHP
21	TAP	M 16.	16.	2.	20.	0.8	0

SNO	TOOL	NOM-φ	NO.	HOLE-φ	HOLE-DEP	PRE-DIA	PRE-DEP	RGH	DEPTH	C-SP	FR	M	M
1	CTR-DR	20.		16.	↓	↓	↓	90	↓	14	0.2		
2	DRILL	14.2	A	14.2	31.	↓	↓	DRILL	T31.	18	0.179		
3	TAP	M 16.	A	16.	20.	↓	↓	FIX	P2.	6	2.		

FIG	PTN	Z	X	Y	AN1	AN2	T1	T2	F	M	N	P	O	R
1	SQR	0.	-70.	-50.	0.	90.	140.	100.	0	2	2	0	0	0

In the present program, the 4th axis is indexed using not the manual program mode unit, but a unit in the basic coordinate system.

Example of Cutting a Cylindrical Cam

To prepare a program to cut the cylindrical cam illustrated.
Take point 0 as the zero point.
First of all, prepare a common unit and an WPC unit.

UNO	MAT	INITIAL-Z	MULTI MODE	MULTI	PITCH-X	PIYCH-Y
0	FCD	50.				

UNO	UNIT	X	Y	Θ	Z	4
1	WPC- 0	-250.	-253.	0.	-269.	0.

Then, use the manual program mode unit to prepare a cutting program.

SNO	G1	G2	DATA 1	DATA 2	DATA 3	DATA 4	DATA 5	DATA 6	S	M/B
1	0	90								M 38
2									1000M	3
3			Z 20.	X 20.						
4			Z -10.							
5			X -260.	4-1260.	F 50.					
6	0		Z 0.							

UNO	UNIT	CONTI.	NUMBER
3	END	1	0

In No. 1 block, select ABSOLUTE and GEAR.
In No. 2 block, input the spindle speed.
In No. 3 block, make a relief 20 mm above the zero point in the basic coordinate system and +20 mm in the positive X direction.

In No. 4 block, lower the cutting edge in the Z-axis direction to where cutting is to be started.

In No. 5 block, move the cutting edge 260 mm in the negative X-axis direction. At the same time, turn the 4th axis 1,260° in the negative direction. In this case, set the feedrate to 50 mm/min.

In this case, the X-axis moves by 260 - (-20) = 280 mm in total.

In the meantime, the rotary table have 3.5 turns = 280 ÷ 80. This may be converted to a right angle as follows:

$$3.5 \times 360 = 1{,}260$$

This 1,260-degree angle is inputted in No. 5 block. F50, is distributed in the 4th axis and in the X-axis in proportion to their strokes.

In No. 6 block, a command is given to return to the basic coordinate system after the cutting edge has separated from the workpiece.

UNO	MAT	INITIAL-Z	MULTI MODE	MULTI	PITCH-X	PITCH-Y
0	FCD	50.				

UNO	UNIT	X	Y	Θ	Z	4
1	WPC- 0	-250.	-253.	0.	-269.	0.

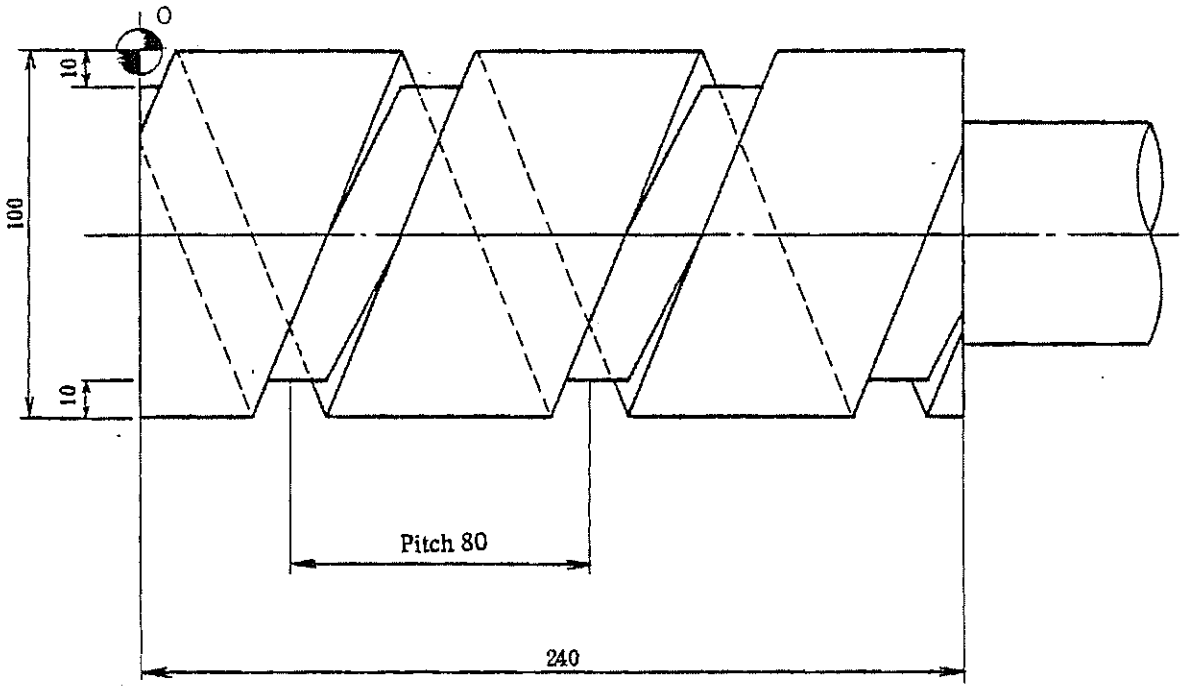
UNO	UNIT	TOOL	NOM-ϕ	NO.
2	MANU PRO	E-MILL	20. A	

SNO	G1	G2	DATA 1	DATA 2	DATA 3	DATA 4	DATA 5	DATA 6	S	M/B
1	0	90								M 38
2	95								1000M	3
3		Z	20.	X 20.						
4		Z	-10.							
5		X	-260.	4-1260.	F 50.					
6	0	Z	0.							

Thus, a cylindrical cam has resulted.

Cutting speeds cannot be specified for the 4th axis independently of the X-, Y- and Z-axes. A ratio of a moving stroke of the 4th axis to that on one of the other three axes can be used to distribute the value of F.
Thus concludes the 4th axis specification.

Example of Cutting a Cylindrical Cam



Cylindrical Cam

MAZATROL M-2 4TH AXIS UNIT (Option)

1. The 4th axis move command can be inputted in the manual program mode unit only.

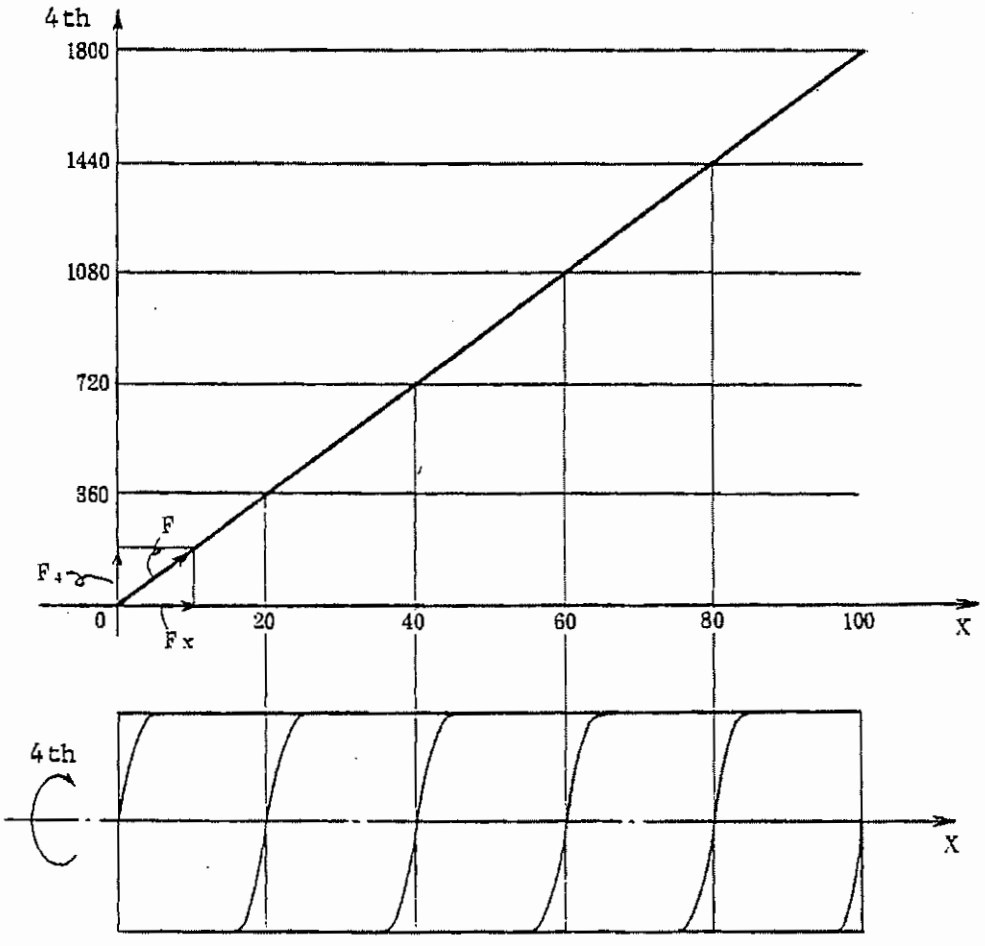
2. A workpiece coordinate system can be set up by inputting data on 4th axis in the WPC unit. Generally, 0 is inputted.

3. Three of X, Y, Z and 4th axes can be interpolated linearly (G01) at a time. However, it is impossible to interpolate an arc which includes the 4th axis (G02 and G03) or to make a helical cut on a workpiece.

4. The linear interpolation including the 4th axis is as illustrated below.

G 0 1 G 9 1 X 1 0 0 4 1800 F 2 0 0

Specify an angle



5. F in the program represents the combined speed of velocity components for each axis.

$F = \sqrt{F_x^2 + F_y^2 + F_a^2}$ (when X, Y and 4th axes are interpolated at a time)

$F_x : F_y : F_a$ = X axis moving stroke : Y axis moving stroke : 4th axis moving stroke