

PROGRAMMING MANUAL

— FOR

Mazak

MAZATROL

CAM T-2

(PRACTICAL USE)

SERIAL NUMBER :

INTRODUCTION

The MAZATROL CAM T-2 is a conversational type of CNC equipment. It was developed by updating the advanced electronics of and introducing certain advanced machining know-how to the MAZATROL CAM T-1, a 2-axis lathe NC developed originally by Yamazaki Machinery Works, Ltd.

The new MAZATROL CAM T-2 is menu-driven and can quickly be automatically programmed to contain machining information through interaction with the NC.

Its color graphics function enables the checking of the actual machining status on the display including checking for the prepared program, tool paths, etc. This PRACTICAL USE MANUAL describes the items needed for preparing programs for each machining mode and tool path.

We are certain that the user will further improve the working efficiency by reading this manual carefully and fully understanding the preparation and operation procedure.

<p>The specifications for the present machine are subject to change without notice for improvement purposes.</p>
--

CONTENTS

	Page
INTRODUCTION	
1. MACHINING PRODUCTS	1-1
1-1 Machine Coordinate System	1-2
1-2 Work Coordinate System	1-3
1-3 Program Coordinate System	1-4E
2. PROGRAM CONFIGURATION	2-1
2-1 Program Configuration	2-2
2-1-1 Common data	2-2
2-1-2 Process data	2-2
2-1-3 Sequence data	2-3
2-1-4 Program end data	2-3
2-2 Configuration of Programs for Several Workpieces	2-3
2-2-1 Number of work programs	2-3
2-2-2 Number of program blocks	2-4E
3. CUTTING PROGRAM DEFINITION AND TOOL PATH	3-1
3-1 Common Data	3-1
3-2 MTR: Material Shape Data	3-4
3-3 BAR: Bar Machining Data	3-6
3-3-1 BAR process data	3-6
3-3-2 BAR sequence data	3-11
3-3-3 Tool path in BAR OUT rough cutting	3-21
3-3-4 Tool path in BAR OUT finish cutting	3-23
3-3-5 Tool path in BAR OUT rough cutting	3-24
3-3-6 Tool path in BAR FCE rough cutting	3-26
3-3-7 Tool path in BAR IN rough cutting	3-27
3-4 CPY: Copy Machining	3-28
3-4-1 CPY process data	3-28
3-4-2 CPY sequence data	3-32
3-4-3 Tool path in CPY IN rough cutting	3-35

	Page
3-5 CNR: Corner Machining	3-37
3-5-1 CNR process data	3-37
3-5-2 CNR sequence data	3-39
3-5-3 Tool path in CNR OUT rough cutting	3-42
3-6 EDG: Edge Machining	3-43
3-6-1 EDG process data	3-43
3-6-2 EDG sequence data	3-45
3-6-3 Tool path in EDG	3-47
3-7 THR: Threading	3-48
3-7-1 THR process data	3-48
3-7-2 THR sequence data	3-53
3-7-3 Tool path in THR OUT and THR OUT	3-56
3-8 GRV: Groove Machining	3-58
3-8-1 GRV process data	3-58
3-8-2 GRV sequence data	3-63
3-8-3 Tool path in GRV OUT and GRV OUT For #0 ...	3-69
3-8-4 Tool path in GRV OUT and GRV OUT For #1, Rough cutting No. 1	3-71
3-8-5 Tool path in GRV OUT and GRV OUT For #1, Rough cutting No. 2	3-72
3-8-6 Tool path in GRV OUT and GRV OUT For #1, Finish cutting	3-74
3-9 DRL: Drilling	3-75
3-9-1 DRL process data	3-75
3-9-2 DRL sequence data	3-78
3-9-3 Tool path for DRL #2	3-80
3-10 TAP: Tapping	3-81
3-10-1 TAP process data	3-82
3-10-2 TAP sequence data	3-83
3-10-3 Tool path in TAP	3-85
3-11 MNP (Manual Program Mode): Manual Program Machining .	3-86
3-11-1 MNP process data	3-86
3-11-2 MNP sequence data	3-90
3-11-3 Tool path in MNP	3-95

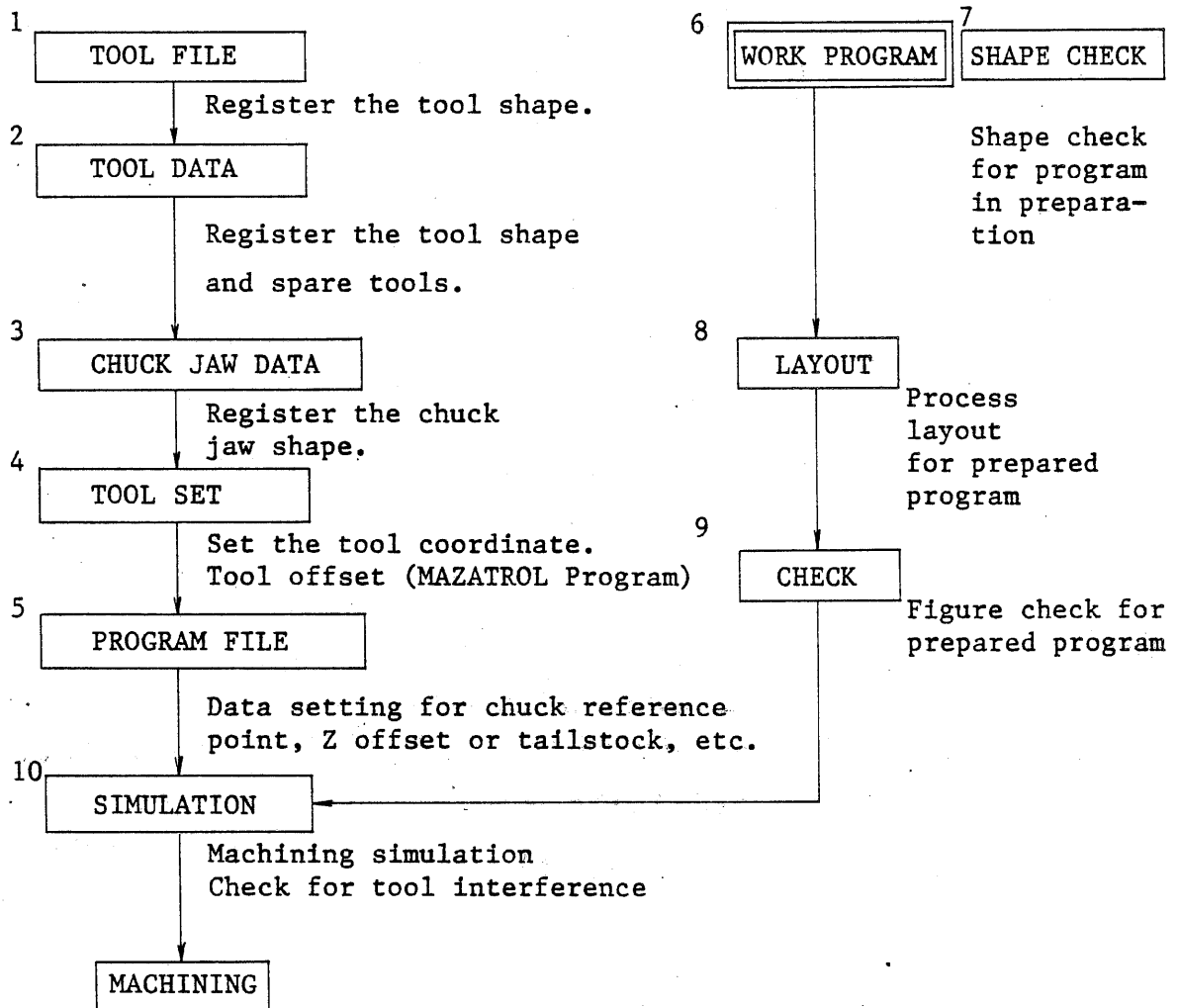
	Page
3-12 M (M Function): M Code	3-97
3-12-1 M process data	3-97
3-12-2 M code table	3-99
3-13 Program End Data	3-106
4. SIMPLE PROGRAM EXAMPLE AND TOOL PATH	4-1
5. PROGRAM EXAMPLE IN CUTTING WORK	5-1
6. FUNCTIONS	6-1
6-1 Automatic Machining Condition Determining Function and Material Registration Function	6-1
6-1-1 Elements determined by CUTTING CONDITION picture	6-3
6-1-2 Elements determined by program data and parameters	6-4
6-1-3 Method of registration of machining conditions	6-6
6-2 Program Edit Function	6-9
6-2-1 PROCESS NO. SEARCH	6-10
6-2-2 LAST SEARCH	6-11
6-2-3 PROCESS COPY	6-12
6-2-4 PROGRAM COPY	6-16
6-2-5 INSERT	6-18
6-2-6 ERASE	6-19
6-3 LAYOUT FUNCTION (Available in MAZATROL Programs only)	6-21
6-3-1 Displaying the LAYOUT picture	6-21
6-3-2 LAYOUT operation	6-23
6-4 Automatic Intersect Point Computation Function	6-28
6-4-1 Intersect point computation for convex and concave arc	6-28
6-4-2 Intersect point computation for taper, GRV sequence	6-31

	Page
7. APPENDIX	7-1
7-1 Various Thread Formulas and Principles	7-1
7-2 Graphs Showing the Relations between Depth of Cut and Number of Cutting Cycles for Drilling (DRL) ...	7-16
7-3 Reducing Depth of Cut	7-17
7-4 Cutting Force and Cutting Power	7-18
7-5 Selecting a Range of Spindle Gear Speeds	7-20
7-6 Nose Shape Compensation	7-22
(1) Compensating for nose shape	7-22
(2) Nose radius offset	7-23
7-7 Simultaneous 2-axis Approach and Return	7-24E

1. MACHINING PRODUCTS

Both the preparation of programs for machining special product shapes and the setup of operations by registering data in the NC in advance are required.

This manual explains the items which are not described in the PROGRAMMING MANUAL "PRIMER".



From TOOL FILE to Actual Machining

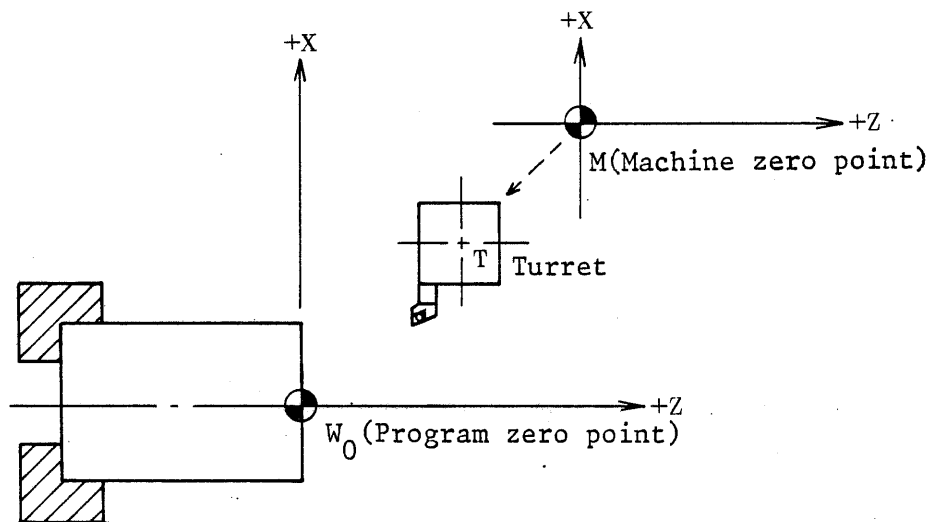
Coordinate System

It is necessary to consider the coordinate system when programming.

The coordinate system includes the machine coordinate system for defining the position of the turret, viewed from a reference point of the machine and the work coordinate system for defining the position of the edge of the tool, viewed from a work reference point.

1-1 Machine Coordinate system (Refer to the Operating Manual for the zero-return method.)

This is to define the turret position when viewed from the machine zero point. The turret position viewed from a machine zero point is indicated in the MACHINE of the COMMAND picture.



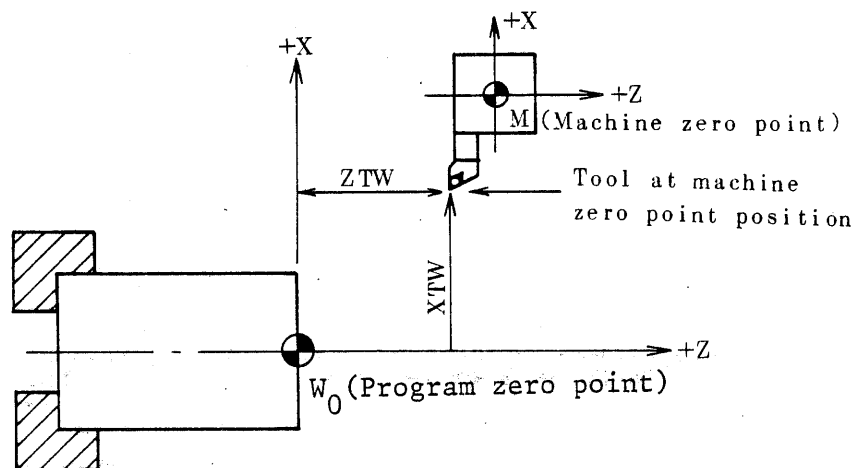
1-2 Work Coordinate System (Refer to the Operating Manual for the tool setting method.)

By setting in the TOOL SET picture the distance to the reference point (W_0) of the work, measured from the tool nose at the machine zero point of the respective tools, the work coordinate system regarding the reference point (W_0) as the work coordinate zero point is established.

Therefore, the program data will be entered by taking (W_0 : program zero point) as the reference point.

In other words, the program zero point must be set carefully in the drawing.

This position is displayed in the "POSITION" on the COMMAND picture.

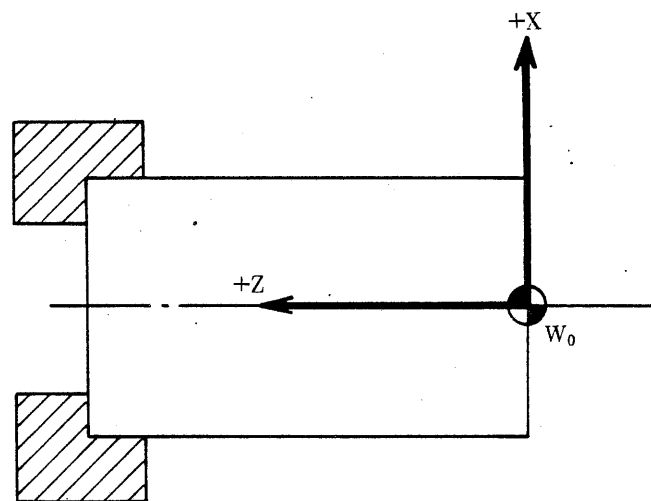


ZTW : Z-axis direction tool set value
 XTW : X-axis direction tool set value
 (radius value)

1-3 Program Coordinate System

Machining dimensions generally are specified by the distance from a reference point.

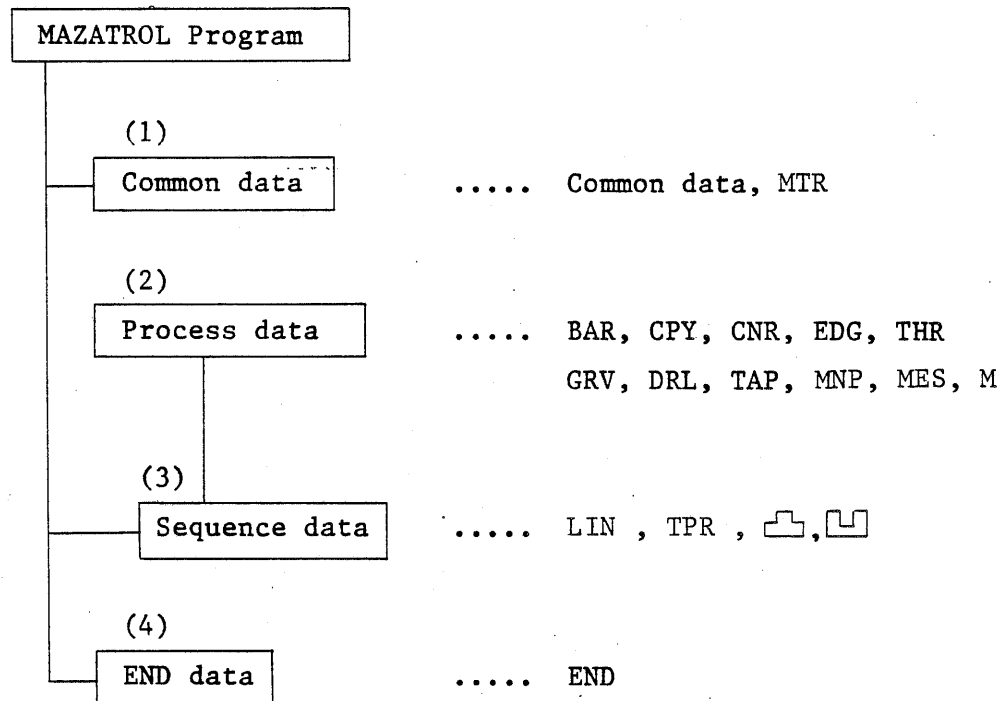
In MAZATROL programs, the program zero point is set at this reference point (usually at the reference edge). The dimensions from the zero point are set in absolute coordinates.



- Note :
- (1) In edge machining mode (EDG), the coordinate in the Z direction above is different. See the edge machining mode explanation.
 - (2) In MAZATROL Programs (except MNP and MES), there are no $-X$ or $-Z$ direction coordinates.

2. PROGRAM CONFIGURATION

MAZATROL Programs are prepared according to the following configuration:



2-1 Program Configuration

2-1-1 Common data

There are two kinds of common data: material data indicated by " PNO 0" and material shape definition data indicated by " PNO 1" MTR which are set after " PNO 0".

" PNO 0" material data are used to set the maximum O.D., minimum I.D., length, spindle maximum r.p.m., finishing allowance and workpiece end amount for the material to be machined. The material data are to be displayed automatically at the head of the program.

The " PNO 1" material shape data: MTR are used to define the arbitrary shape of material (e.g. forged material). These data can only be set after " PNO 0" but cannot be set after any other " PNO". More than two sets of material shape definition data cannot be set within one program.

These data are used in common to prepare programs.

2-1-2 Process data

Process data are indicated by " PNO ", which shows the tool operation mode and associated data.

Process data include:

- Ⓐ BAR: Bar material machining
- Ⓑ CPY: Copy machining
- Ⓒ CNR: Corner machining
- Ⓓ EDG: Edge machining
- Ⓔ THR: Threading
- Ⓕ GRV: Groove machining
- Ⓖ DRL: Drilling
- Ⓗ TAP: Tapping
- Ⓘ MNP: Manual program mode
- Ⓙ MES: Measure
- Ⓚ M : M code

They are selected from the menu. The number of process data which can be specified in the program is maximum 99. A maximum of 100 process data including the common data can be registered. If this number is exceeded, the alarm 417 "PROCESS DATA > 99" is given.

2-1-3 Sequence data

Sequence data are indicated by "SEQ", and relate to the machining dimensions on a drawing.

A several sets of sequence data can be specified only for bar machining (BAR), copy machining (CPY), threading (THR) and manual program mode (MNP).

Maximum 20 sequence data can be specified within one process for bar machining (BAR), copying (CPY) and threading (THR). For manual program mode (MNP), maximum 250 sequence data can be specified. If these numbers are exceeded, the alarm 419 "NO. OF SEQUENCES > 20" or alarm 414 "NO. OF SEQUENCES > 250" is given.

2-1-4 Program end data

This data shows the action to be taken when machining is completed. One program end data is specified at the end of a program.

2-2 Configuration of Programs for Several Workpieces

Programs for several workpieces can be registered beforehand so that machining may be performed according to the work Nos.

2-2-1 Number of work programs

Maximum 16 work programs can be registered. If more than 17 programs are registered, the alarm 413 "NO. OF PROGRAMS > 16"

is given. Work Nos. must be specified for each program when it is prepared.

2-2-2 Number of program blocks

A maximum of 500 blocks can be stored. If the number of blocks to be stored is greater than 500, the alarm 404 "NO. OF BLOCKS IN A PRGM > 500" is given. Similarly, a maximum of 500 blocks can be registered in one program.

3. CUTTING PROGRAM DEFINITION AND TOOL PATH

3-1 Common Data

PNO	MAT	OD-MAX	ID-MIN	LENGTH	RPM	FIN-X	FIN-Z	WORK FACE
0	a.	b.	c.	d.	e.	f.	g.	h.

a. "WORKPIECE MATERIAL (MENU)?"

To allow the automatic selection of machining conditions, select the kind of material from the menu and input it.

CBN STL	ALY STL	CST IRN	ALUMINUM	STAINLS				
---------	---------	---------	----------	---------	--	--	--	--

CBN STL : Carbon steel for machine structure

ALY STL : Chromium-molybdenum steel

CST IRN : Gray cast iron

ALUMINUM: Aluminium or aluminium alloy

STAINLS : Stainless steel

The kind of material which is not found on the menu should be set in terms of the material No. shown on CUTTING CONDITION NO. 2 picture.

As data, the percentage (%) of each cutting conditions registered on the CUTTING CONDITION NO. 1 picture shall be entered on the CUTTING CONDITION NO. 2 picture.
(Refer to 6. FUNCTION.)

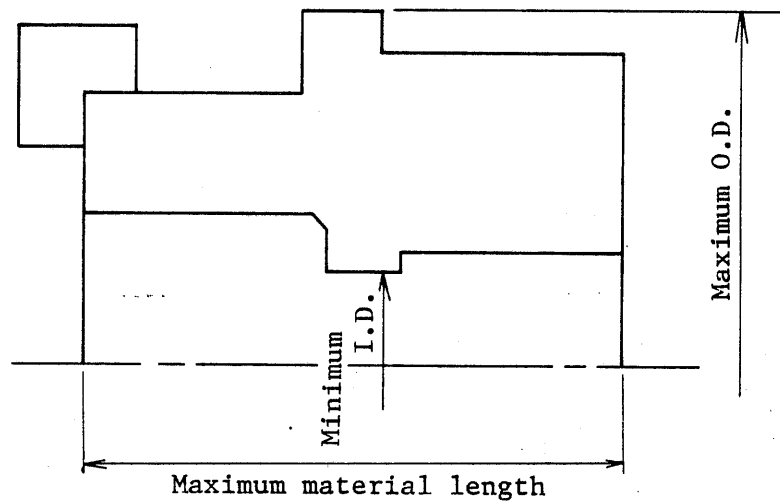
b. "MAX OUTER DIA. OF WORKPIECE?"

Set the maximum O.D. of the material (in mm).

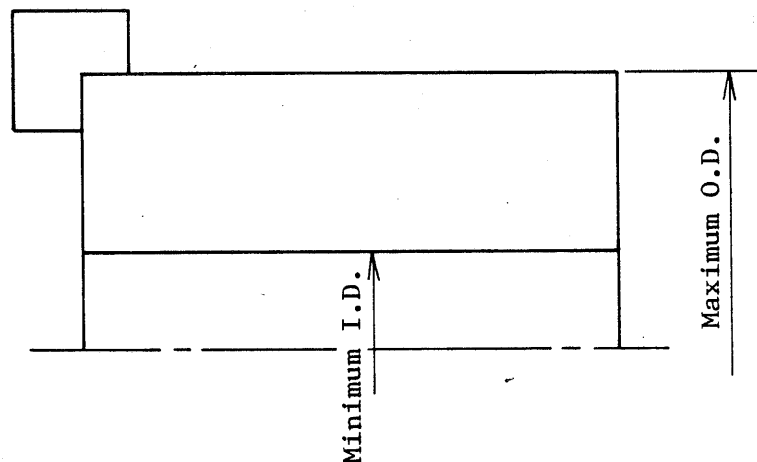
c. "MIN INNER DIA. OF WORKPIECE?"

Set the minimum I.D. of the material (in mm). If the I.D. is not to be machined, set "0".

Molded material



Bar material



The maximum O.D. and minimum I.D. are used to define the tool swing position and to define the safe tool path in approach from the tool swing position to the cutting starting point and in return of the tool from the cutting final point to the tool swing position.

d. "WORKPIECE LENGTH?"

Set the maximum length of the material (in mm).

e. "MAX. SPINDLE RPM LIMIT?"

Set the maximum value of spindle r.p.m. which assures safety. The r.p.m. is not limited if "0" has been set.

f. "FINISH ALLOWANCE-X?"

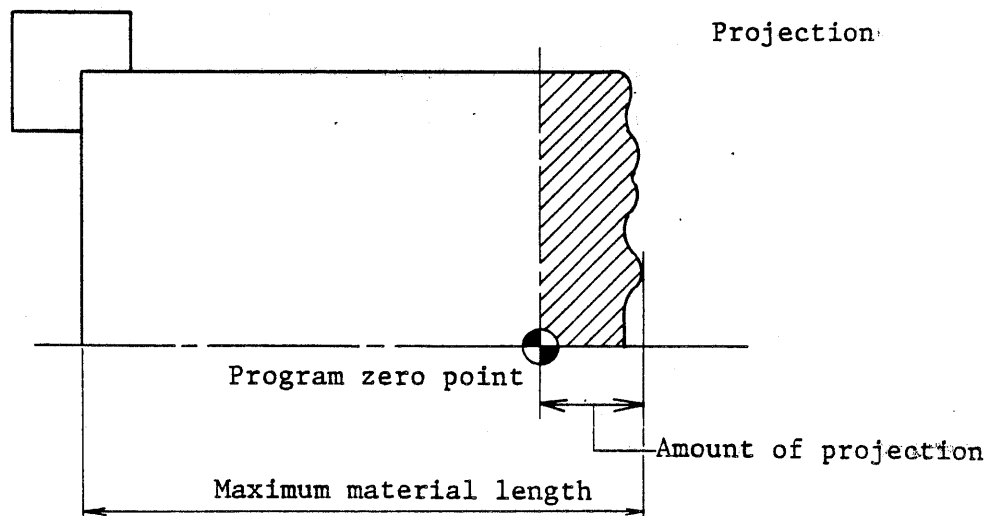
The finishing allowance to be cut with the finishing tool in the X direction shall be set in terms of the value of diameter (in mm). This value is applicable to bar machining (BAR), copy machining (CPY), and corner machining (CNR) in common.

g. "FINISH ALLOWANCE-Z?"

Set the finishing allowance (in mm) in the Z direction to be cut with the finishing tool. This value is applicable to bar machining (BAR), copy machining (CPY), edge machining (EDG) and corner machining (CNR) in common.

h. "STOCK REMOVAL OF WORK FACE?"

Set the projection amount at the face of the material (in mm).



The projection is not regarded as material in any mode other than edge machining mode, therefore, proceed to the next process after cutting off the projection in the edge machining mode (EDG).

Note : (1) Maximum material O.D., I.D., or material length can be set up to the third lower digit (in mm)/fourth lower digit (in inch) than the decimal point but they are indicated on the picture to the first lower digit (in mm)/second lower digit (in inch) than the decimal point. (Maximum 6 digits can be inputted.)

(2) When data is wrongly set for common data, the alarm 317 "ILLEGAL MATERIAL ID/OD", alarm 318 "ILLEGAL MATERIAL LENGTH" are given. Also, the program containing only the manual program process causes an alarm.

3-2 MTR: Material Shape Data

This data is used to set the shape of materials (e.g. forged materials) other than round bars. The material shape of a round bar material is, as mentioned above, defined by common data specifying the maximum O.D., I.D. and length.

Select the WORKPIEC SHAPE key from the menu.

PNO	MODE	X1/4	Z1/4	X2/5	Z2/5	X3/6	Z3/6
*	MTR OUT	a.	a.	b.	b.	c.	c.
		d.	d.	e.	e.	f.	f.
	IN	g.	g.	h.	h.	i.	i.

- a. "WORKPIECE OUTER DIA. P1-X?"
"WORKPIECE OUTER DIA. P1-Z?"
- b. "WORKPIECE OUTER DIA. P2-X?"
"WORKPIECE OUTER DIA. P2-Z?"
- c. "WORKPEICE OUTER DIA. P3-X?"
"WORKPIECE OUTER DIA. P3-Z?"
- d. "WORKPIECE OUTER DIA. P4-X?"
"WORKPIECE OUTER DIA. P4-Z?"
- e. "WORKPIECE OUTER DIA. P5-X?"
"WORKPIECE OUTER DIA. P5-Z?"
- f. "WORKPIECE OUTER DIA. P6-X?"
"WORKPIECE OUTER DIA. P6-Z?"

a. through f.

Enter the O.D. shape points. A maximum of six points can be entered as O.D. but not all need be entered. (Unit: mm)

- g. "WORKPIECE INNER DIA. P1-X?"
"WORKPIECE INNER DIA. P1-Z?"
- h. "WORKPIECE INNER DIA. P2-X?"
"WORKPIECE INNER DIA. P2-Z?"
- i. "WORKPIECE INNER DIA. P3-X?"
"WORKPIECE INNER DIA. P3-Z?"

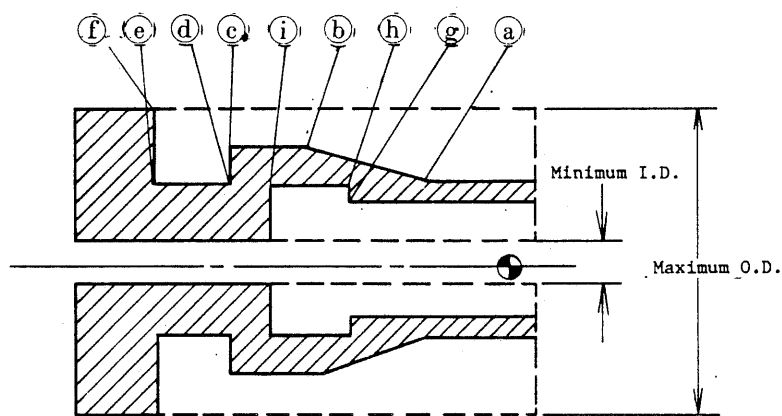
g. h. and i.

Enter the I.D. shape points. A maximum of three points can be entered but not all need be. (Unit: mm)

Notes: (1) MTR (material shape) can be set after common data.
More than two MTRs can not be set within one program.


(2) (X, Z) = (0,0) is invalid.

(3) Both O.D. and I.D. shapes can be set up to the third lower digit (in mm)/fourth lower digit (in inch) than the decimal point but they are indicated on the picture up to the first lower digit (in mm)/second lower digit (in inch).



3-3 BAR : Bar Machining Data

The BAR machining data is used to machine a bar material.

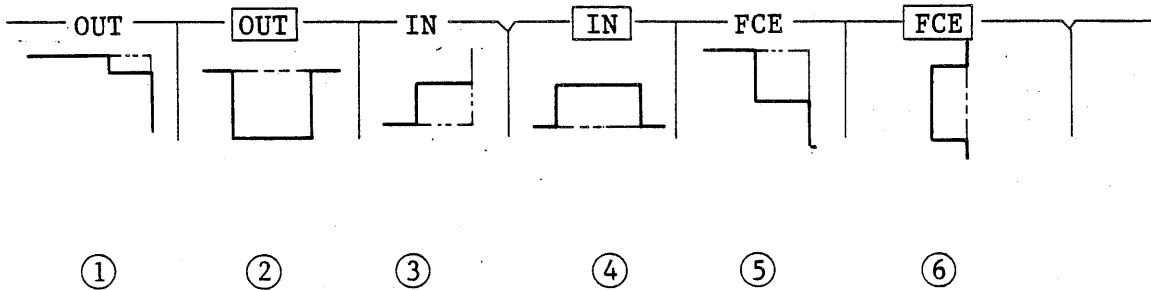
Select the  key on the menu.

3-3-1 BAR process data

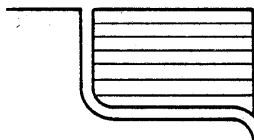
PNO	MODE	PART	CPT-X	CPT-Z	RV	FV	FEED	DEPTH	GR	RT	FT
*	BAR	a.	b.	c.	d.	e.	f.	g.	h.	i.	j.

a. "MACHINING PART (MENU)?"

Select the area to be machined from the menu. There are six bar machining patterns:



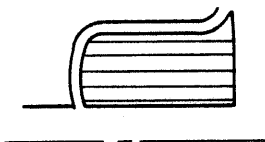
① O.D. open



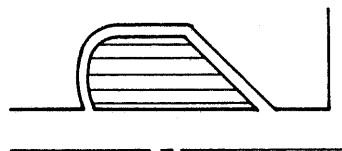
② O. D. halfway



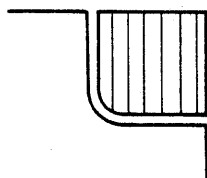
③ I.D. open



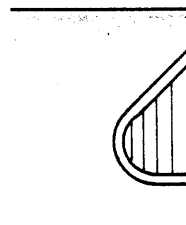
④ I.D. halfway



⑤ Front open



⑥ Front halfway



Select a pattern applicable to each portion to be machined.

b. "CUTTING POINT-X?"

Enter the point of cutting with the tool in the X direction by the diameter value (in mm).

c. "CUTTING POINT-Z?"

Set the point of cutting with the tool in the Z direction (in mm).

The cutting points as well as sequence data define the area of cutting. If the material shape data (MTR) is set, the cutting area beyond the shape is not regarded as material.

d. "SURFACE SPEED FOR ROUGH CUT?"

Set the surface speed for rough cutting (in m/min.).

Select the AUTO ? key on the menu, and the following are also determined automatically:

d. RV e. FV f. FEED g. DEPTH

o BAR mode roughing surface speed determining element:

CUTTING CONDITION picture

(BAR : RV (m/min) * (MATERIAL code (%))

These values can be changed freely and they can be set directly.

e. "SURFACE SPEED FOR FINISH CUT?"

Set the surface speed for finish cutting. (in m/min)

o BAR mode finish surface speed

determining element: CUTTING CONDITION picture

(BAR: FV (m/min)) * (MATEREIAL code (%))

In case the automatically determined values are to be used, move the cursor to the next position, using the cursor key or INPUT key.

f. "FEEDRATE (/REV) FOR ROUGH CUT?"

Set the amount of feed (in mm/rev.) per one revolution of the spindle for rough cutting.

- o BAR mode rough cutting feedrate per revolution
determining element: CUTTING CONDITION picture

The value of (BAR : FEED (mm/rev.)) is entered.

g. "DEPTH OF CUT?"

Set the maximum amount of cut per path in rough cutting with the diameter value (single side thickness) (in mm).

Maximum amount of cut in BAR mode rough cutting: CUTTING CONDITION picture

The value of (BAR : DEPTH (mm)) is entered.

h. "GEAR NO.?"

Set the spindle gear used in rough cutting. When "0" is set, the gear for the r.p.m. calculated from the minimum diameter and surface speed (RV) in that process is automatically selected.

If any gear has been specified, that gear is selected accordingly. In finish machining, the gear No. is automatically determined by the surface speed (FV) whether the gear No. or "0" may be specified. This gear No. concept applies to the other processes except manual program mode (MNP).

i. "ROUGHING TOOL NO. (MENU)?"

Set the tool No. used for rough cutting.

The TOOL DATA key on the menu is selected to confirm the tool Nos. registered on the tool data picture. If rough cutting is not to be performed, select the NONE key on the menu or push the INPUT key or cursor key to move to the next position.

This concept of the ROUGHING TOOL NO. and FINISHING TOOL NO. also applies to the other processes.

"OFFSET NO.?"

Set the offset No. of the tool for rough cutting.

Each tool has a pair of offset Nos. in each of the X and Z directions. Set "1" or "2" if an offset is desired. A figure greater than 3 cannot be entered. If "0" has been set no offset is executed. "1" and "2" mean "OFFSET-1" and "OFFSET-2" on the TOOL SET picture. Enter the amounts of offset here.

This OFFSET No. concept also applies to the other processes.

j. "FINISHING TOOL NO.?"

Set the tool No. used for finish cutting.

Perform setting in the manner similar to i. above.



"OFFSET NO.?"

Set the offset No. of tool for finish cutting in the manner similar to i. above.

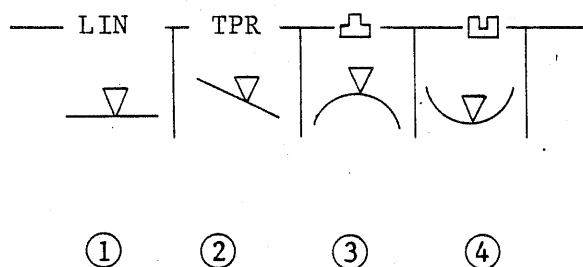
3-3-2 BAR sequence data



SEQ	SHP	S-CNR	SPT-X	SPT-Z	FPT-X	FPT-Z	F-CNR	RADIUS	SRF	\$	M
	a.	b.	c.	c.	d.	d.	e.	f.	g.	h.	i.

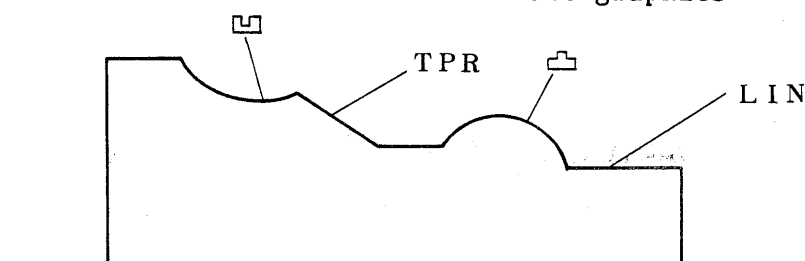
a. "SHAPE PATTERN (MENU)?"

Using the menu, select the patterns (straight line: LIN, taper: TPR, convex arc: , concave arc: ) constituting the shape and center of the arc (CENTER) used for calculations of the intersect point.

There are four shape patterns:

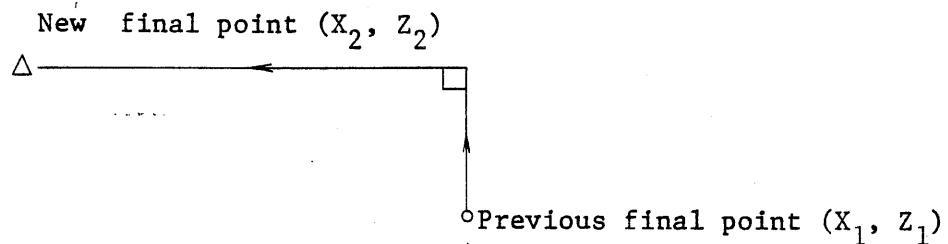


- ① LIN (straight line) ... Used to enter linear graphics
- ② TPR (taper) Used to enter tapered graphics
- ③  (convex arc) Used to enter convex arc graphics
- ④  (concave arc) Used to enter concave arc graphics



① LIN (straight line)

To enter a straight line, only the final point (d. FPT-X, FPT-Z) may be specified (in mm). Then, a straight line heading for the final point is formed. Two lines forming a right angle are automatically produced between the final point in the new graphics and that in the old graphics.



② TPR

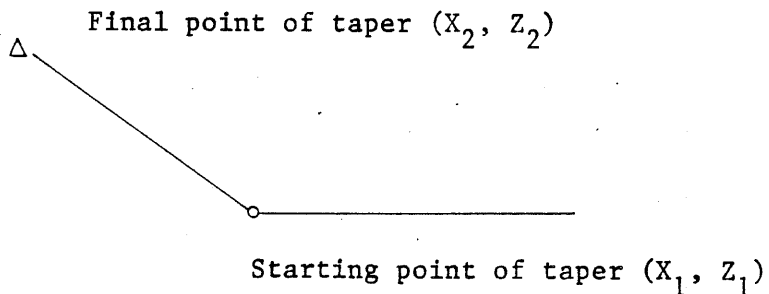
TPR

Taper : (1) Starting and final points are known.

(2) Either of starting or final point is unknown, and taper angle is known.

(1) Starting and final points are known:

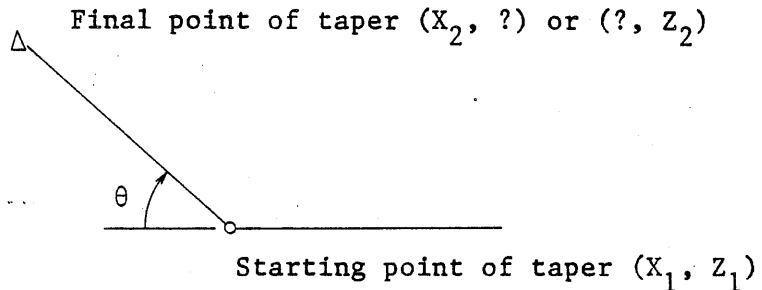
Enter the starting point (c. SPT-X, SPT-Z) and final point (d. FPT-Z, FPT-Z) and final point (d. FPT-Z, FPT-X) of taper.



(2) Either of starting or final point is unknown and taper angle θ is known.

Enter the starting point (c. SPT-X, SPT-Z) and the final point (d. FPT-X or FPT-Z) of taper and the taper angle f .

Ex.) When final point is unknown:



- o If the starting point of a taper graphics begins with the final point of the previous graphics, push the `CONTINUE` key on the menu. Then, the values of the final point of previous graphics are automatically entered as the coordinates of the starting point of the new graphics.

In case CENTER (center of the arc) is entered before TPR sequence data, however, the values of the final point of the graphics prior to the previous one are entered automatically.

- o If X or Z of the starting or final point is unknown, push the `?` key on the menu.
- o The taper angle may be entered as a negative value. The positive and negative taper angles at each portion are determined as shown below:

Positive taper angle			
Negative taper angle			
	O.D. Cutting (OUT)	I.D. cutting (IN)	Face cutting (FCE)

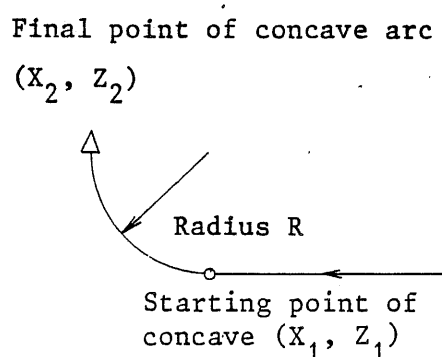
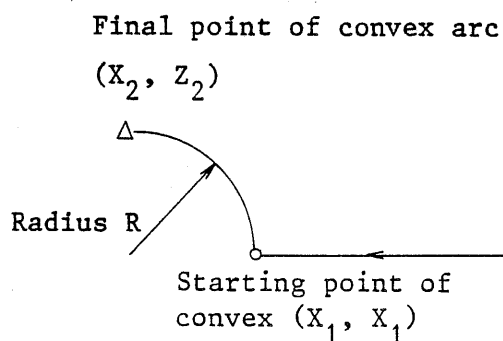
The arrows in the above views indicate the cutting directions.

③ (Convex arc)

A convex arc is defined by specifying the starting point (c. SPT-X, SPT-Z), final point (d. FPT-X, FPT-Z) and radius (f. RADIUS R) of the arc.

④ (Concave arc)

A concave arc is defined by specifying the starting point (c. SPT-X, SPT-Z), final point (d. FPT-X, FPT-Z) and radius (f. RADIUS R) of the arc.



- o If one of the intersect points between the arc and a straight line or taper is unknown when the center of the convex or concave arc and its radius is known, the intersect points can be found by calculation by the automatic intersect point computation function.

b. "STARTING CORNER?"

c. "STARTING POINT-X?" "STARTING POINT-Z?"

Set the starting point position for taper, convex arc and concave arc.

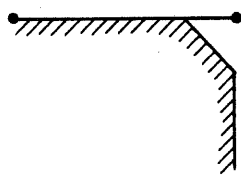
d. "FINAL POINT-X?" "FINAL POINT-Z?"

Set the final point position in each pattern.

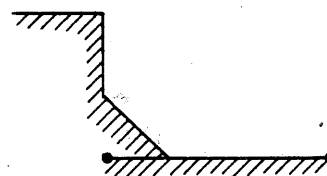
e. "FINAL CORNER?"

The area at the starting point of the graphics element is called the starting corner and that at the final point is called the final corner. These corners can be chamfered or rounded.

Final point Starting point



Starting corner



Final point Starting point

Final corner

1. Corner chamfering

If values are put in the STARTING CORNER and FINAL CORNER, chamfering becomes possible.

2. Corner rounding

If a value is entered with CORNER R on the menu inverted, chamfering is possible with this value as arc radius.

f. "RADIUS R?" or "ANGLE?"

Set the taper angle or the radius for convex or concave arc.

g. "FINISH SURFACE ROUGHNESS (MENU)?"

Select the finished surface roughness code from the menu. This causes the feed speed for the finishing process to be calculated automatically.

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

(EX.) Nose R: 0.8mm

Surface roughness code	1	2	3	4	5	6	7	8	9
Surface roughness (μm)	100	50	25	125	6.3	3.2	1.6	0.8	0.4
Feedrate (min./rev.)	0.8	0.565	0.4	0.282	0.2	0.143	0.101	0.071	0.05

$$F = \sqrt{\frac{8R\mu}{1000}} \text{ mm/rev}$$

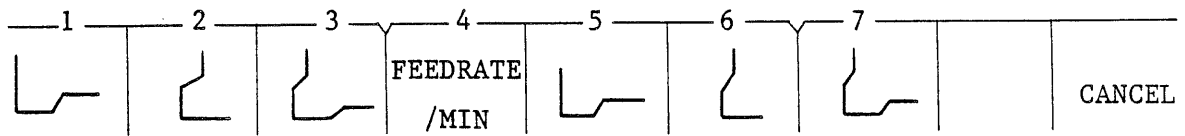
F: Feedrate

R: Nose R (mm)

μ: Surface roughness (μm)

h. "NECKING OR FR/MIN (MENU)?"

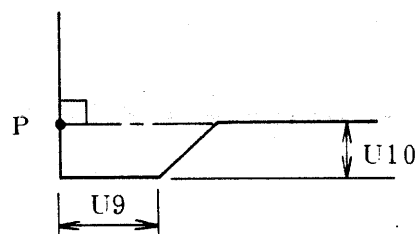
Set the shape of the grinding allowance or feedrate per minute. Select the grinding allowance shape code or feedrate per minute from the menu.



	O.D. cutting	I.D. cutting	Face cutting
Forward direction			
Reverse direction			

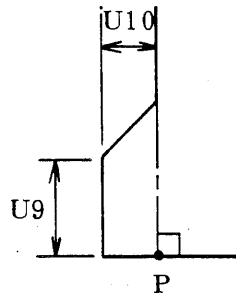
The grinding allowance shape is set by parameters.
(P in the drawings represents the final point.)

Grinding allowance code 1



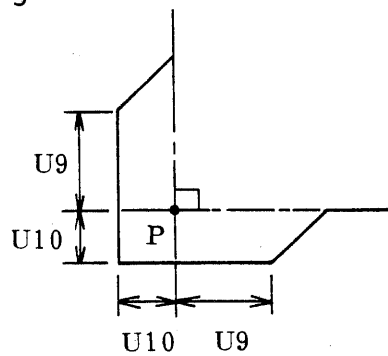
User parameter U9, U10
(X direction stands for the radius.)

Grinding allowance code 2



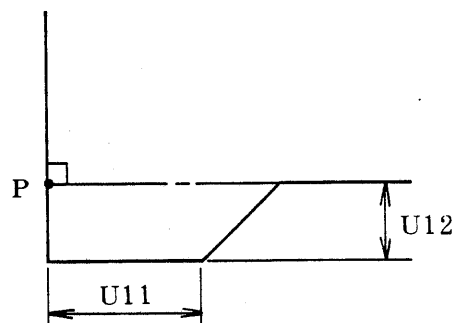
User parameter U9, U10
(X direction stands
for the radius.)

Grinding allowance code 3



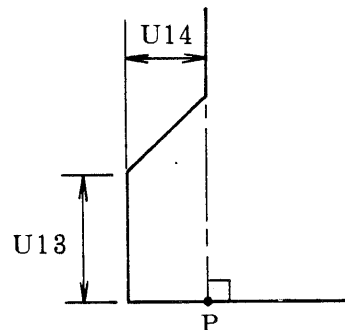
User parameter U9, U10
(X direction stands
for the radius.)

Grinding allowance code 5



User parameter U11,
U12 (X direction
stands for the
radius.)

Grinding allowance code 6

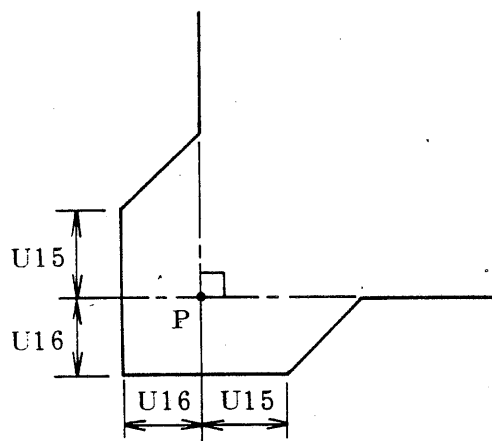


User parameter U13,
U14 (X direction
stands for the
radius.)

Grinding allowance code 7

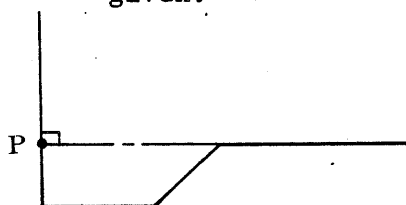
User parameter **U15**,

U16 (X direction stands for the radius.)



The finish allowance shape must be determined on condition that:

- 1) The terminating end portion of the straight line should form a right angle to the next adjoining straight line.
- 2) The tool tip angle should be limited when the grinding allowance is given.

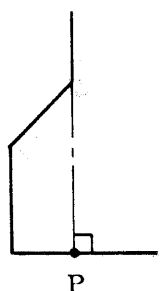


Grinding allowance code Cutting angle $A \geq 93^\circ$

1 and 5

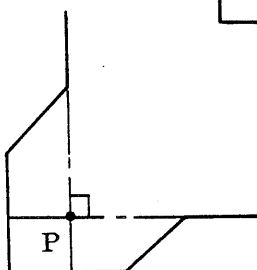
Tool tip angle $B \leq 60^\circ$

$A + B \leq 153^\circ$



Grinding allowance code Cutting angle $A \geq 117^\circ$

2 and 6

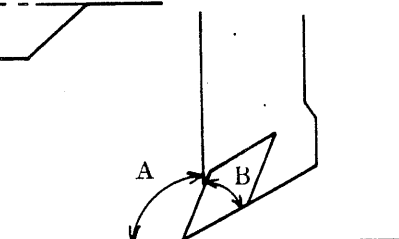


Grinding allowance code Cutting angle $A \geq 117^\circ$

3 and 7

Tool tip angle $B \leq 36^\circ$

$A + B \leq 153^\circ$



A : cutting angle

B : tool tip angle

If the grinding allowance code 4 has been set, the feedrate per minute becomes effective. It is used in combination with g. SRF to control the feedrate. (See the table below.)

Roughness code	0	1	2	3	4	5	6	7
Allowance code	4	4	4	4	4	4	4	4
Feedrate (mm/min.)	2,000	1,000	720	520	370	270	200	140
Roughness code	8	9						
Allowance code	4	4						
Feedrate (mm/min.)	100	72						

1. "M CODE (MENU)?"

The M code can be set by the menu keys or ten keys.

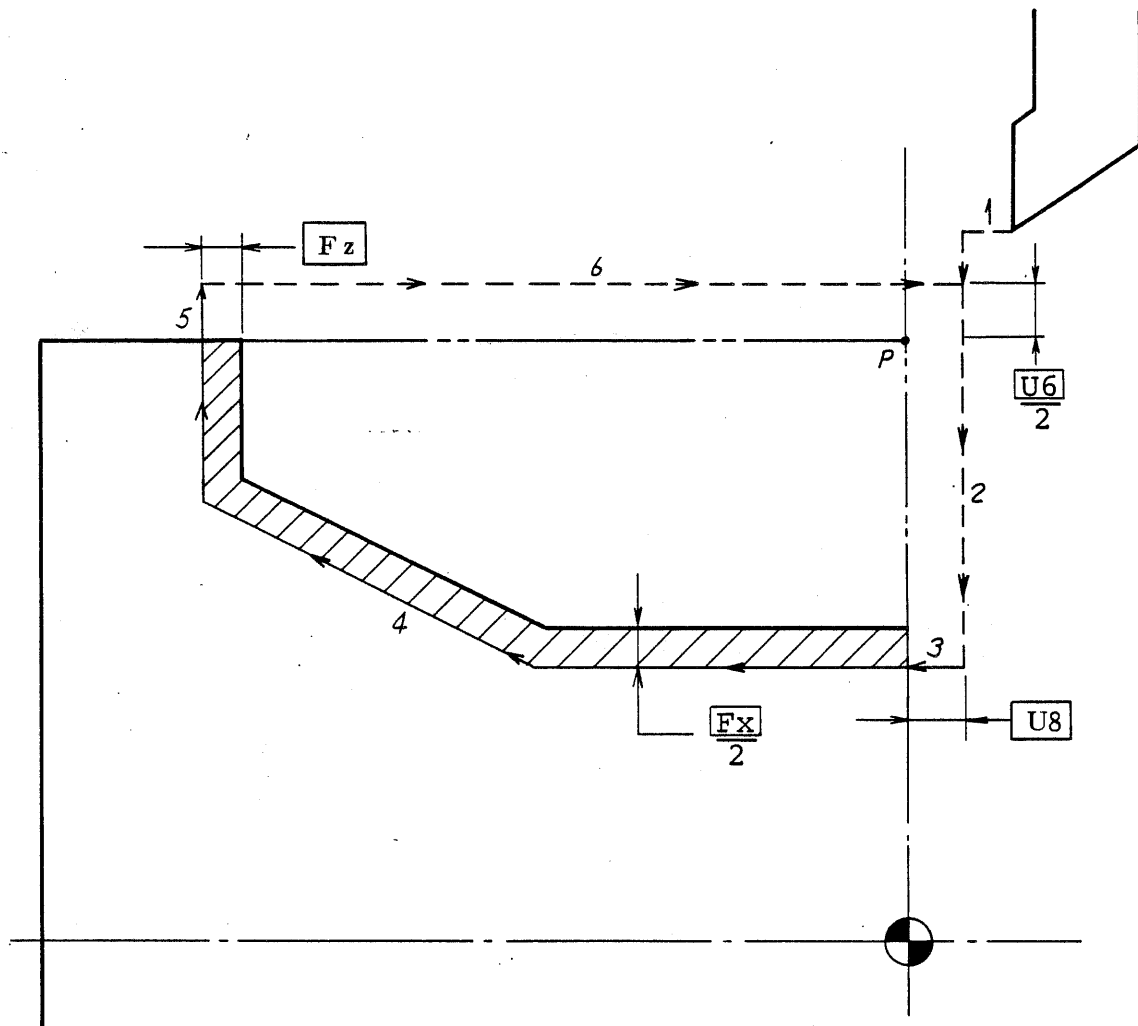
This is an auxiliary function.

M functions similar to those in the M mode of the process data can be entered. M functions becomes effective in the finish process mode and on. To use the coolant from the rough machining, set the M functions with the process data in the M mode. For the M codes, refer to M (M FUNCTION) in process data.

Note: When the program zero point and either of the coordinate of the starting point-X or Z, final point-X or Z of the sequence data becomes similar, machining in some cases may not be performed.

- (2) As compared with other tool paths, speed is increased in the tool path 24 portion.
- Select "1" (valid) or "2" (invalid) for the speed increase at the bar roughing cycle wall, using the parameter and set the increase rate with the parameters and . The speed increase is effective only for the last block of the last wall during bar roughing cycle machining.
- (3) If the shape of roughing in the MAZATROL program is a wall, the tool is set apart from the wall by the amount equivalent to the clearance set by the parameter before it is moved. (See "Parameter".)

3-3-4 Tool path in BAR OUT finish cutting



P : CPT-X, Z set by program process data

F_x : FIN-X set by program common data " PNO 0"

F_z : FIN-Z set by program common data " PNO 0"

U6: Safety contour clearance (O.D.) set by parameter U6

U8: Safety contour clearance (face) set by parameter U8

U60: Amount of relief from the wall in final rough cutting set by the parameter **U60** .

Notes: (1) The tool paths 8, 14, 20 and 26 are the same.

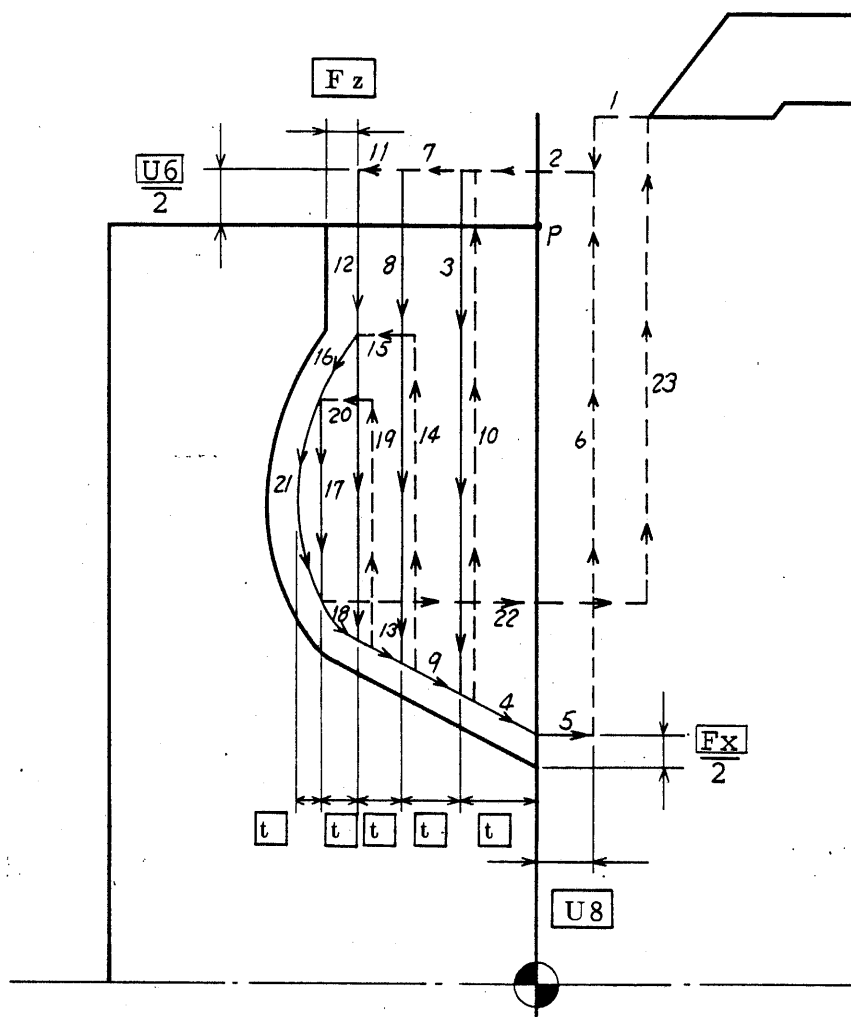
(2) Feedrate increases in the tool path 29 portion.
The parameter **U23** is used to determine whether the feedrate increase should be valid or invalid. The parameter **U56** and **U57** are used to set the increase rate.

(3) Halfway machining (including but not limited to BAR **OUT**)
The square portion (indicated by oblique lines) as shown in the figure cannot be machined. This is because the cutting is done with an allowance of 3° with respect to the tool tip angle. The same applies to the other portions to be machined.

$$\theta_1 = \theta_2 - 3^\circ$$

Thus, to cut the portion indicated by the oblique lines, CNR (corner machining) is used.

3-3-6 Tool path in BAR FCE rough cutting



P : CPT-X, Z set by program process data

t_1 : DEPTH (first cut) set by program process data

F_x : FIN-X set by program common data "PNO 0".

F_z : FIN-Z set by program common data "PNO 0"

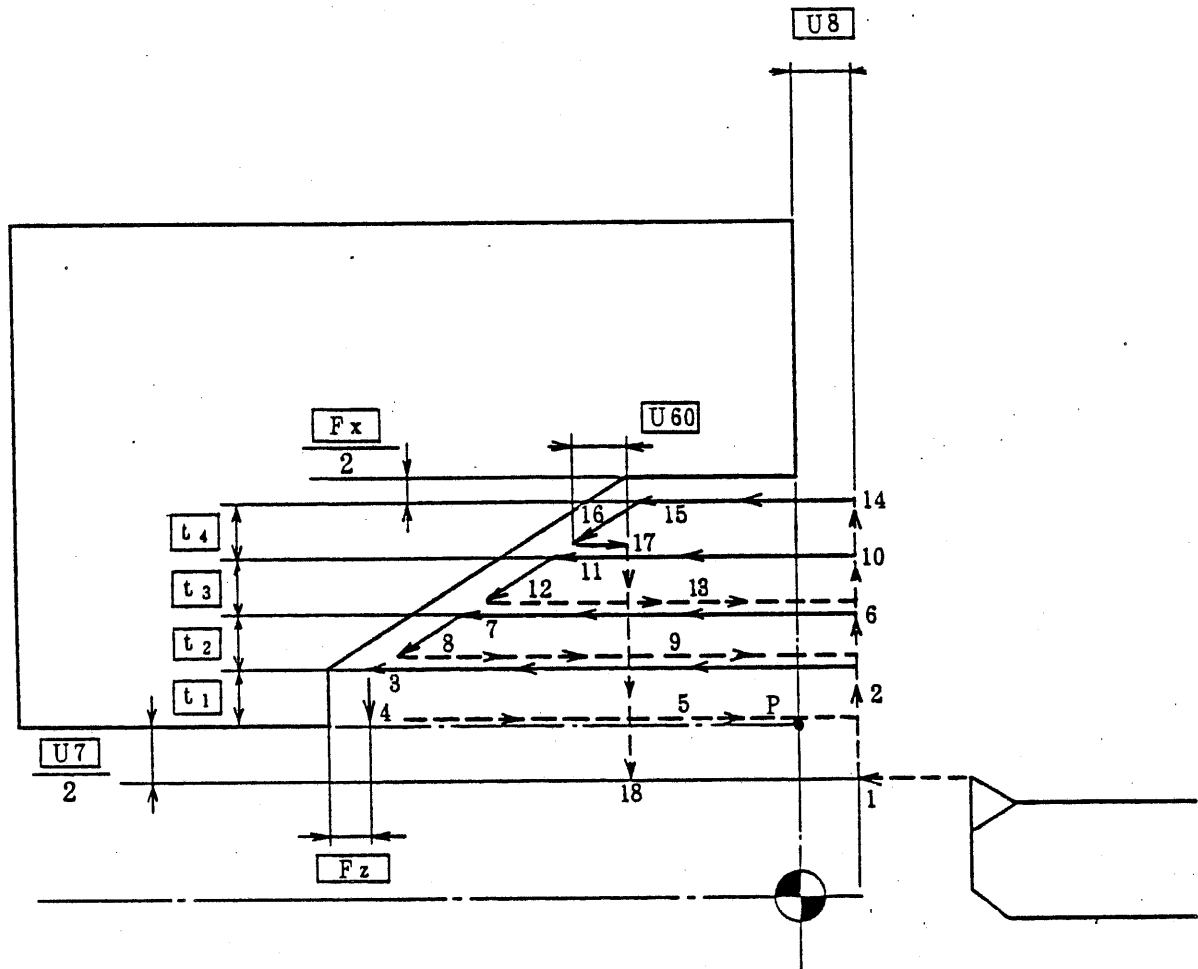
U6: Safety contour clearance (O.D.) set by parameter U6.

U8: Safety contour clearance (face) set by parameter U8.

Notes (1) The tool paths 3, 8 and 12 are the same as tool paths 10, 14 and 19, respectively.

(2) As in BAR OUT roughing cycle, the feedrate increases at the tool path 21 portion. The parameter U23 is used to determine whether feedrate should be increased or not. The feedrate increase rate is set by the parameter U56 and U57.

3-3-7 Tool path in BAR IN rough cutting



P : CPT-X,Z set by program process data

t_1 : DEPTH (first cut) set by program process data

t_1 through t_4 : Amount of cut into which cutting amount decrease rate set by the parameter $U59$ is incorporated.

Fx : FIN-X set by program common data "PNO 0".

Fz : FIN-Z set by program common data "PNO 0".

U7 : Safety contour clearance (I.D.) set by parameter $U7$

U8 : Safety contour clearance (face) set by parameter $U8$

U60 : Amount of relief from the wall in final rough cutting set by parameter $U60$

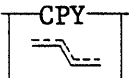
Notes: (1) The tool paths 3 and 7 are the same as tool paths 9 and 13, respectively.

- (2) Feed rate is increased in the tool path 16 portion.

Select valid or invalid for the speed increase using the parameter U23 and set the increase rate with the parameters U56 and U57.

3-4 CPY : Copy Machining

This mode is used when machining forged and similar materials.

Select  from the menu.

3-4-1 CPY process data

PNO	MODE	PART	CPT-X	CPT-Z	SRV-X	SRV-Z	RV	FV	FEED	DEPTH	GR	RT	FT
*	CPY	a.	b.	c.	d.	e.	f.	g.	h.	i.	j.	k.	l.

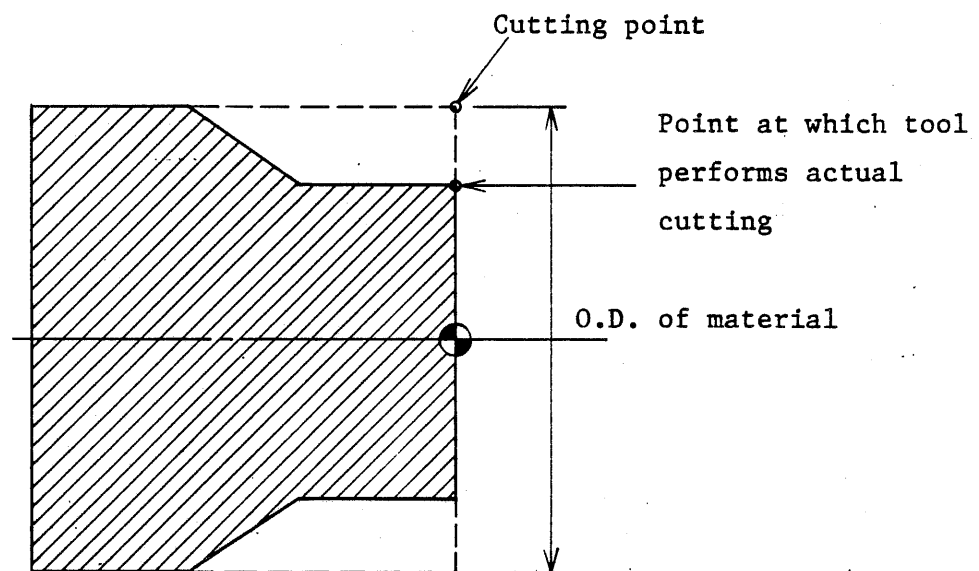
- a. "MACHINING PART (MENU)?"

Select the machining range from the menu.
The same patterns as BAR are used in CPY.

- b. "CUTTING POINT-X?"

c. "CUTTING POINT-Z?"

The cutting point means the point at which the tool actually machines. In copy machining:CPY, the cutting point is defined as shown in the following figures.
(Unit: mm)



d. "STOCK REMOVAL-X?"

e. "STOCK REMOVAL-Z?"

Set the cutting allowance at the thickest portion of the copied material (in mm).

f. "ROUGHING SURFACE SPEED?"

Set the surface speed for rough machining. (in m/min)

Select the AUTO ? key on the menu, and f. RV, g. FV,

h. FEED and i. DEPTH are automatically determined.

- o CPY mode rough machining surface speed determining element: CUTTING CONDITION picture

(CPY : RV (m/min)) * (MATERIAL code (%))

These values can be changed freely. They can also be set directly.

g. "SURFACE SPEED FOR FINISH CUT?"

Set the surface speed for finish machining (in m/min.).

- o CPY mode finish machining surface speed determining element: CUTTING CONDITION picture

(CPY : FV (m/min)) * (MATERIAL code (%))

h. "FEEDRATE (/REV) FOR ROUGH CUT?"

Set the feedrate per revolution of the spindle for rough machining. (Unit: mm/rev.)

- o CPY mode rough machining feedrate per revolution determining element: CUTTING CONDITION picture

The value of (CPY : FEED (mm/rev)) is entered.

i. "DEPTH OF CUT?"

Set the maximum cut per path in rough machining as the radius value (single side thickness) (in mm).

- o Maximum amount of cut in CPY mode rough machining determining element: CUTTING CONDITION picture

The value of (CPY : DEPTH (mm)) is entered.

j. "GEAR NO.?"

Set the spindle gear No. in roughing.

When nothing has been specified or "0" is set, the gear is selected automatically.

k. "ROUGHING TOOL NO.?"

Set the tool No. for roughing.

"OFFSET NO.?"

Set the offset No. of the tool for roughing.

l. "FINISHING TOOL NO.?"

• Set the tool No. for finishing.

"OFFSET NO.?"

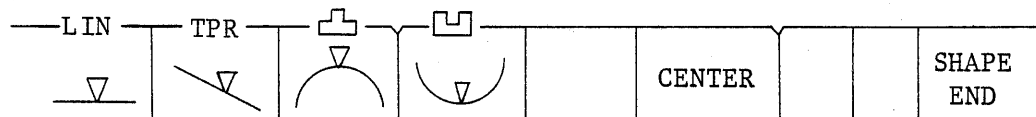
Set the offset No. of the tool for finishing.

3-4-2 CPY sequence data

SEQ	SHP	S-CNR	SPT-X	SPT-Z	FPT-X	FPT-Z	F-CNR	RADIUS	SRF	\$	M
*	a.	b.	c.	c.	d.	d.	e.	f.	g.	h.	i.

a. "SHAPE PATTERN (MENU)?"

Select the center of the arc used for patterns as with bar material (BAR) to define the shape and for calculations of the intersect points, using the menu.



For the meaning of each shape, refer to "BAR".

b. "STARTING CORNER?"

e. "FINAL CORNER?"

The starting and final corner can be faced or rounded.

c. "STARTING POINT-X?" "STARTING POINT-Z?"

Set the starting point position in each taper, convex arc or concave arc.

If the starting point of a taper, convex arc or concave arc coincides with the final point of the previous linear sequence, push the CONTINUE on the menu. Then, the value will be entered automatically. If either the starting point X or Z of a taper, convex arc or concave

arc is unknown, push the $\overline{?}$ key on the menu to have the intersect point calculated on the CRT.

d. "FINAL POINT-X?" "FINAL POINT-Z?"

Set the final point position of each pattern shape. If either the final point X or Z for a convex or concave arc is unknown, push the $\overline{?}$ key on the menu to have the intersect point automatically calculated on the CRT.

f. "RADIUS R?" or "TAPER ANGLE?"

Set the taper angle or the radius for a convex or concave arc.

g. "FIN SURFACE ROUGHNESS (MENU)?"

Select the finished surface roughness code from the menu. Then, the feedrate for finishing is automatically determined and set.

∇	∇	$\nabla\nabla$	$\nabla\nabla$	$\nabla\nabla\nabla$	$\nabla\nabla\nabla$	$\nabla\nabla\nabla$	$\nabla\nabla\nabla\nabla$	$\nabla\nabla\nabla\nabla$
1	2	3	4	5	6	7	8	9

h. "NECKING OR FR/MIN (MENU)?"

Set the shape of grinding allowance or feedrate per minute. Select the grinding allowance shape code or feedrate per minute from the menu.

1	2	3	4	5	6	7	
			FEEDRATE /MIN				CANCEL

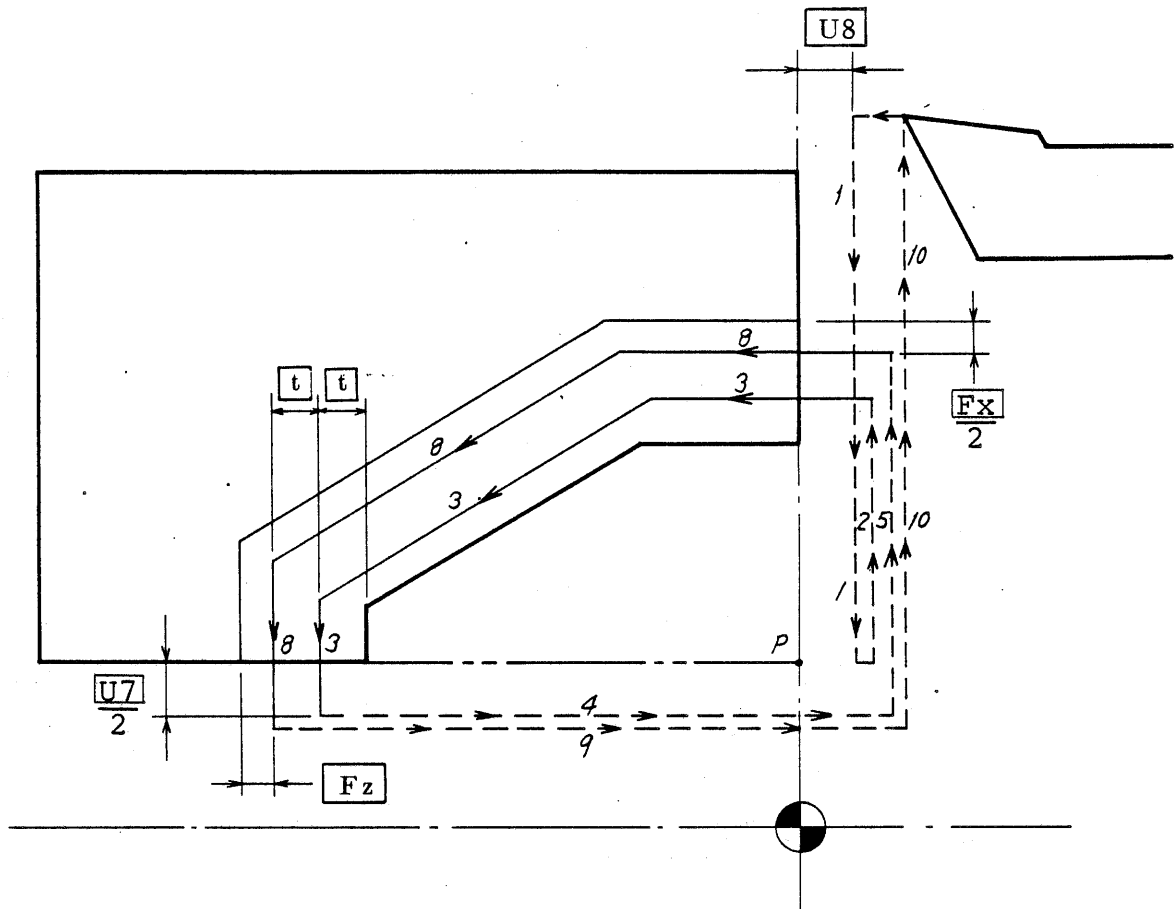
i. "M CODE (MENU)?"

The M code can be set with the menu keys or ten keys.

This is an auxiliary function.

M functions similar to those in the M mode of the process data can be entered. The M functions become effective when the machine is put in the finishing mode. For the M codes, refer to M (M function) in Process Data.

3-4-3 Tool path in CPY IN rough cutting



P : CPT-X, Z set by program process data

t : DEPTH set by program process data

F_x : FIN-X set by program common data " PNO 0"

F_z : FIN-Z set by program common data " PNO 0"

U7: Safety contour clearance (O.D.) set by parameter U7 .

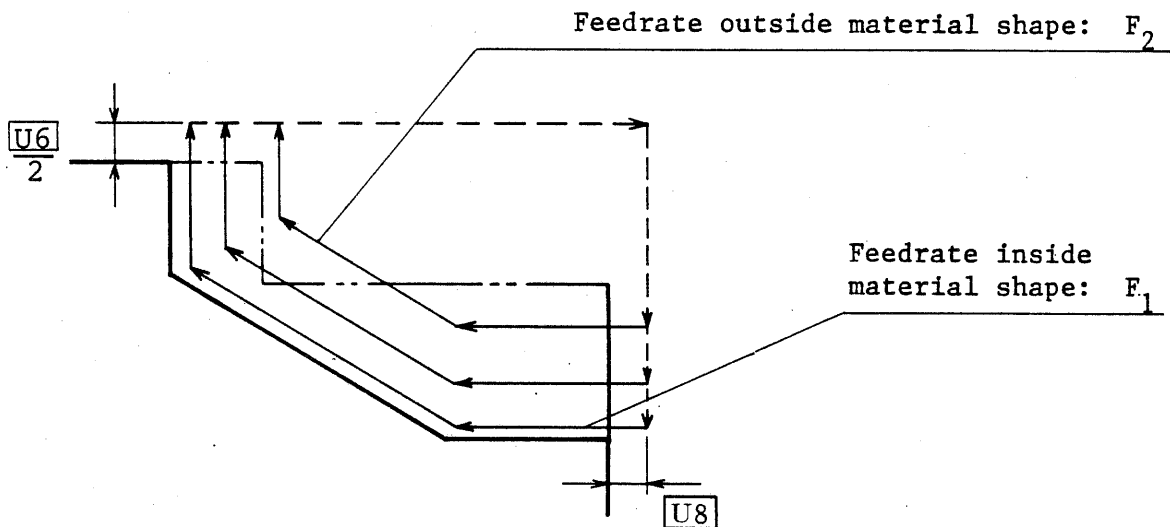
U8: Safety contour clearance (I.D.) set by parameter U8 .

Notes: (1) The tool paths 4, 1 and 5 are the same as 9, 2 and 10, respectively.

- (2) When the tool path has fallen out of the material shape during the CPY roughing cycle, the cutting feedrate in that while can be increased according to the values set in the parameter **U58**.

$$F_2 = F_1 \times \frac{100 + \text{U58}}{100}$$

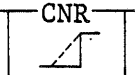
In case of CPY OUT:





3-5 CNR: Corner Machining

Some portions may be left uncut at the corners when cutting is performed halfway. This data is used to remove that portion to make the corner square.

Select the  from the menu.

3-5-1 CNR process data

PNO	MODE	PART	RV	FV	FEED	DEPTH	GR	RT	FT
*	CNR	a.	b.	c.	d.	e.	f.	g.	h.

a. "MACHINING PART (MENU)?"

Select the machining range from the menu. In corner machining, setting is made in the following manner for both open and halfway type cuttings:

OUT = OUT , IN = IN , FCE = FCE

b. "SURFACE SPEED FOR ROUGH CUT?"

Set the surface speed in roughing (in m/min.).

Push the AUTO ? on the menu, and the following will be automatically determined:

b. RV c. FV d. FEED e. DEPTH

- o CNR mode rough cutting surface speed determining element:
CUTTING CONDITION picture

(CNR : RV (m/min.)) * (MATERIAL code (%))

These values can be varied freely. They can also be set directly.

c. "SURFACE SPEED FOR FINISH CUT?"

Set the surface speed in finish cutting. (in m/min)

- o CNR mode finish surface speed determining element:
CUTTING CONDITION picture

(CNR : FV (m/min)) * (MATERIAL code (%))

d. "FEEDRATE (/REV) FOR ROUGH CUT?"

Set the feedrate per revolution of the spindle for roughing (in mm/rev.).

- o CNR mode rough cutting feedrate per revolution determining element: CUTTING CONDITION picture

The value of (CNR : FEED (mm/rev)) is entered.

e. "DEPTH OF CUT?"

Set the maximum amount of cut in rough cutting as the radius value (single side thickness) (in mm).

- o Maximum amount of cut in CNR mode rough cutting determining element : CUTTING CONDITION picture

The value of (CNR: DEPTH (mm)) is entered.

f. "GEAR NO.?"

Set the spindle gear No. in roughing. The gear number is automatically determined if nothing or "0" has been set.

g. "ROUGHING TOOL NO. (MENU)?"

Set the tool No. for roughing.

"OFFSET NO.?"

Set the offset No. of the tool for roughing.

h. "FINISHING TOOL NO.?"

Set the tool No. for finishing.

"OFFSET NO.?"

Set the offset No. of the tool for finishing.

3-5-2 CNR sequence data

SEQ	SPT-X	SPT-Z	FPT-X	FPT-Z	F-CNR	SRF	\$	M
*	a.	a.	b.	b.	c.	d.	e.	f.

a. "STARTING POINT-X?" "STARTING POINT-Z?"

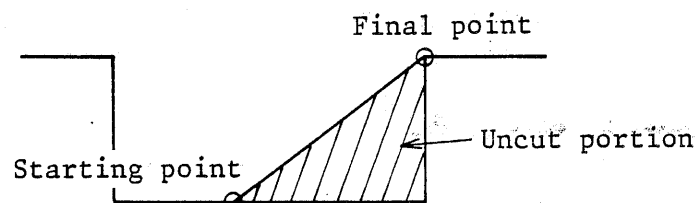
It is used to set the corner machining starting point
(See the figure below.)

b. "FINAL POINT-X?" "FINAL POINT-Z?"

Set the final point in corner machining. (See the figures below.)

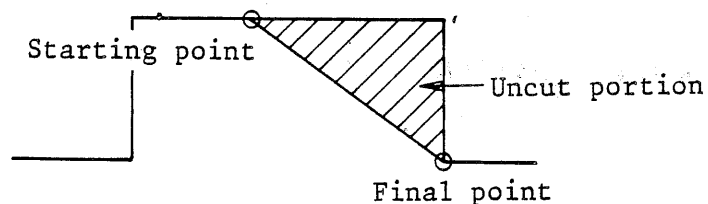
CNR OUT

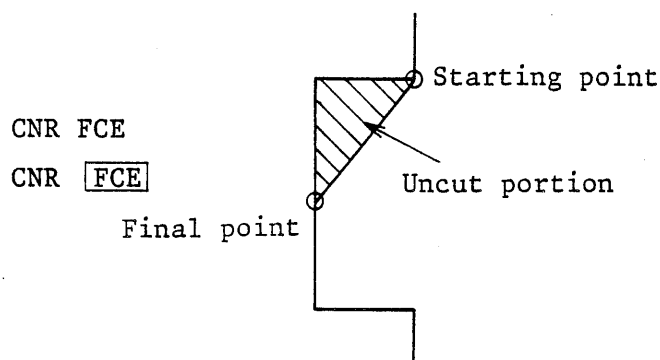
CNR ☐ OUT



CNR IN

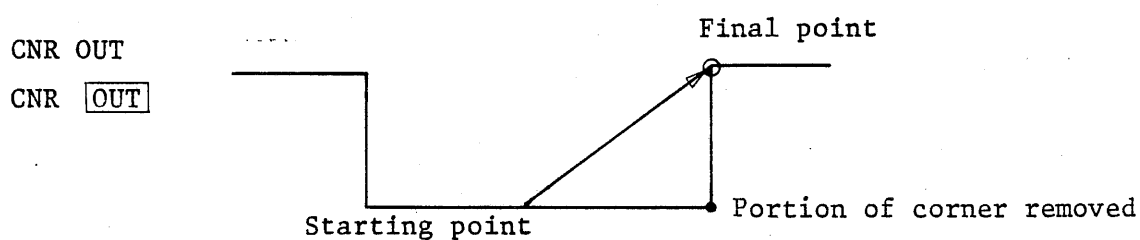
CNR ☐ IN



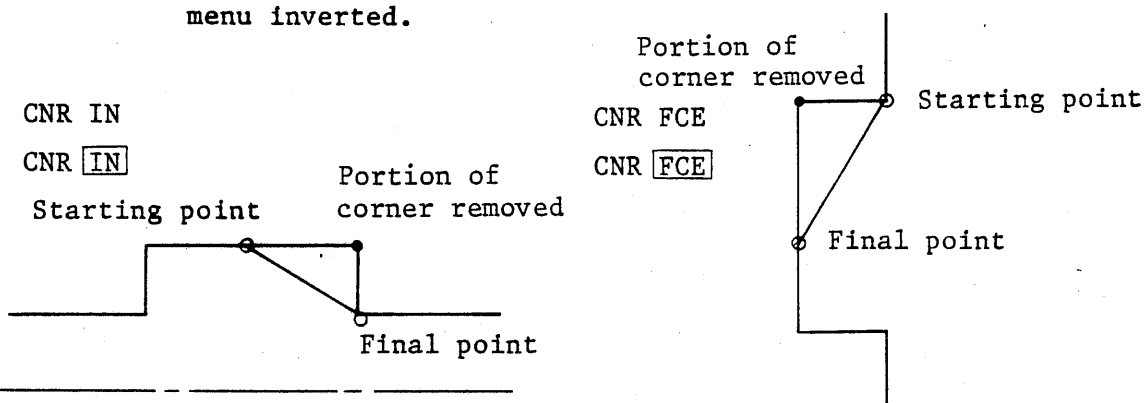


c. "FINAL CORNER?"

It is used for facing and rounding of a corner.



To face the corner, enter true value directly. To round the corner, enter the value with the **CORNER R** on the menu inverted.



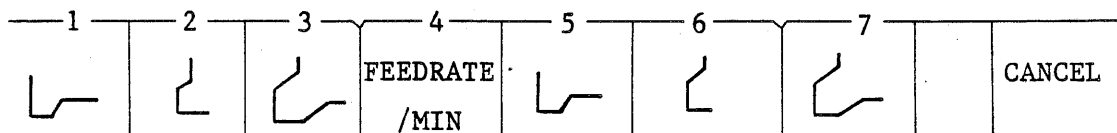
d. "FIN SURFACE ROUGHNESS (MENU)?"

Enter a finished surface roughness code, selecting it from finish codes on the menu. Then, the feedrate in finishing will be automatically calculated and set.

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

e. "NECKING OR FR/MIN (MENU)?"

Set the shape of grinding allowance or feedrate per minute. The grinding allowance is provided at the corner. Select the grinding allowance shape code or feedrate per minute from the menu.

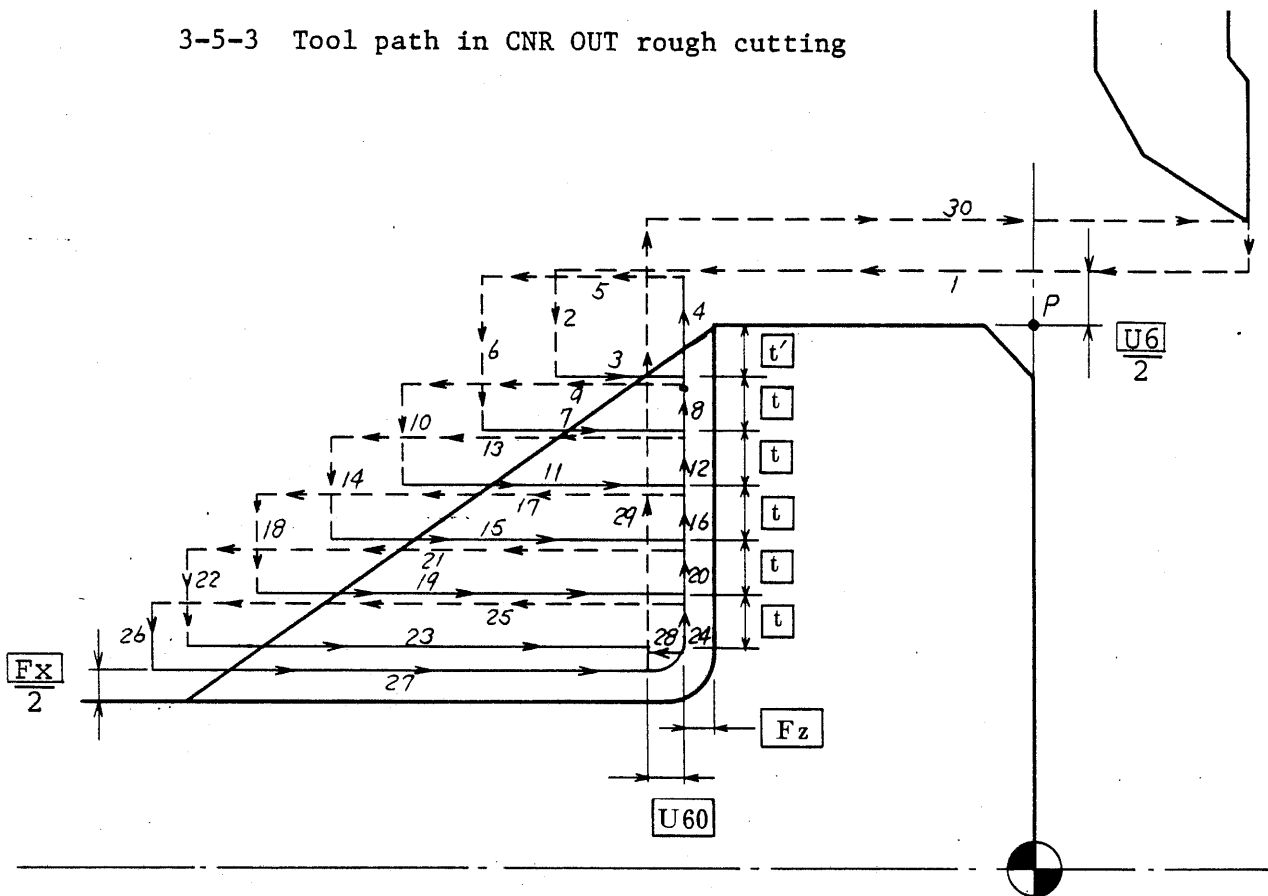


f. "M CODE (MENU)?"

An auxiliary function allows the setting of the M code with meny keys or ten keys.

An M functions as in the M mode of process data can be entered. The M function thus entered becomes effective from the finishing mode and on. For M codes, see M (M function) of process data.

3-5-3 Tool path in CNR OUT rough cutting



P : CPT-X, Z set by program process data

t' : Amount of cut calculated by NC with the DEPTH set by program process data added to the safety contour clearance in parameter U6.

t : DEPTH set by program process data

F_x : FIN-X set by program common data " PNO 0"

F_z : FIN-Z set by program common data " PNO 0"

U6 : Safety contour clearance (O.D.) set by parameter U6

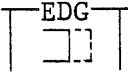
U60 : Amount of relief from wall in final roughing set by parameter U60

Notes: (1) Tool paths 3, 7, 11, 15 and 19 are same as the paths 9, 13, 17, 21 and 25, respectively.

(2) Amount of relief (tool paths 5, 9, 13, 17, 21 and 25) in the Z direction are calculated by NC with the safety contour clearance set in parameter U8 taken into account.

3-6 EDG : Edge Machining

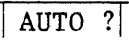
This function is used to machine the projection on the edge.

Select the  on the menu.

3-6-1 EDG process data

PNO	MODE	RV	FV	FEED	DEPTH	GR	RT	FT
*	EDG	a.	b.	c.	d.	e.	f.	g.

a. "SURFACE SPEED FOR ROUGH CUT?"

Set the surface speed in rough cutting (in m/min.). Push the  on the menu, and the following will be determined automatically:

a. RV b. FV c. FEED d. DEPTH

- o EDG mode roughing surface speed determining element :
CUTTING CONDITION picture

(EDG : RV (m/min)) * (MATERIAL code (%))

These values can be changed freely. They can be also set directly.

b. "SURFACE SPEED FOR FINISH CUT?"

Set the surface speed in finishing (in m/min.).

- o EDG mode finish surface speed determining element:
CUTTING CONDITION picture

(EDG : FV (m/min.)) * (MATERIAL code (%))

c. "FEEDRATE (/REV) FOR ROUGH CUT?"

Set the feedrate per revolution of the spindle in roughing (in mm/rev.).

EDG mode rough cutting feedrate per revolution
determining element: CUTTING CONDITION picture

The value of (EDG : FEED (mm/rev.)) is entered as it is.

d. "DEPTH OF CUT?"

Set the maximum amount of cut per path in the Z direction in rough cutting (in mm).

o Maximum amount of cut in EDG mode rough cutting
determining element: CUTTING CONDITION picture

The value of (EDG : DEPTH (mm)) is entered as it is.

e. "GEAR NO.?"

Set the spindle gear No. in rough cutting. If nothing or "0" has been set, the gear number is selected automatically.

f. "ROUGHING TOOL NO.?"

Set the tool No. in rough cutting.

"OFFSET NO.?"

Set the offset No. for the tool used in rough cutting.

g. "FINISHING TOOL NO.?"

Set the tool No. for finish cutting.

"OFFSET NO.?"

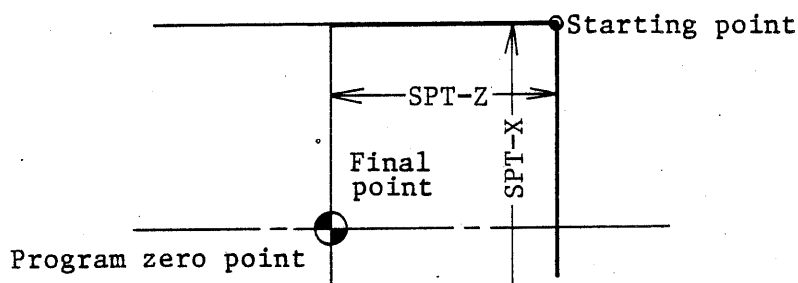
Set the offset No. of the tool used for finish cutting.

3-6-2 EDG sequence data

SEQ	SPT-X	SPT-Z	FPT-X	FPT-Z	SRF	\$	M
*	a.	a.	b.	b.	c.	d.	e.

a. "STARTING POINT-X?" "STARTING POINT-Z?"

Set the cutting start point on the projection of the material edge. Although the starting point is located to the right of the program zero point, the value of the starting point Z is entered as the positive value.



b. "FINAL POINT-X?" "FINAL POINT-Z?"

Set the final point of cutting at the projection on the material edge. (In the above figure, the final point is the program zero point.)

c. "FIN SURFACE ROUGHNESS (MENU)?"

Enter the finish surface roughness code from the menu. Then, the feedrate in finish cutting is automatically calculated and set.

d. "NECKING OR FR/MIN (MENU)?"

If "4" is entered here after setting C. finish surface roughness code, feedrate per minute becomes effective. (Refer to the description of the grinding allowance in BAR sequence data.)

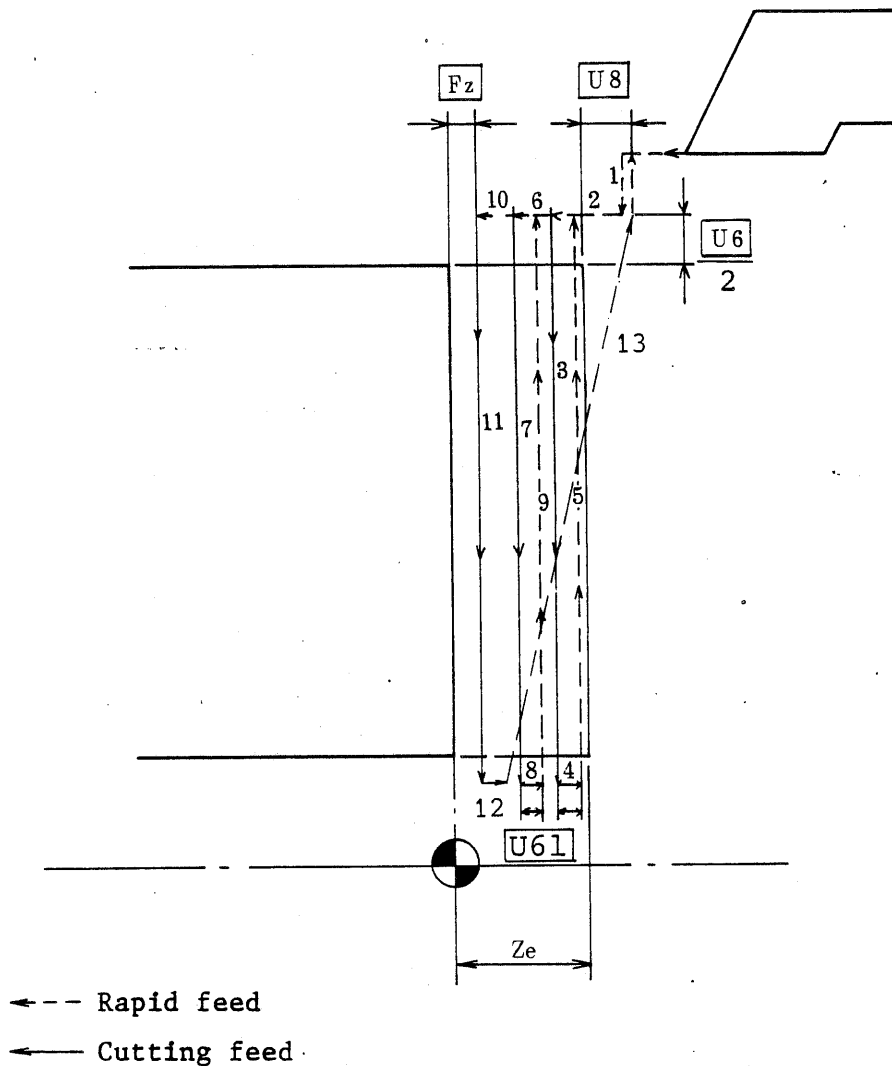
Setting of 1, 2, 3, 4, 5, 6 or 7 of the grinding allowance code, however, is invalid.

e. "M CODE (MENU)?"

An M code can be set with menu keys or ten keys. This function is auxiliary. The M code thus set becomes effective when finish machining begins.

For M codes, see M (M function) of process data.

3-6-3 Tool path in EDG



P : CPT-X, Z set by program process data

Z_e : WORK FACE (amount of projection) set in program common data "PNO 0"

F_z : FIN-Z set in program common data "PNO 0"

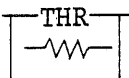
U6: Safety contour clearance (O.D.) set in parameter U6

U8: Safety contour clearance (face) set in parameter U8

U61: Amount of return (parameter U61) after cutting in EDGE rough cutting

This becomes effective on finish machining also.

3-7 THR : Threading

Used for threading. Select the  from the menu.

3-7-1 THR process data

PNO	MODE	PART	CHAMF	LEAD	ANG	MULTI	HGT	NUMBER	V	DEPTH	GR	T
*	THR	a.	b.	c.	d.	e.	f.	g.	h.	i.	j.	k.

a. "MACHINING PART (MENU)?"

Select the range of machining from the menu. In threading, there is no difference between the open type and halfway type cuttings:

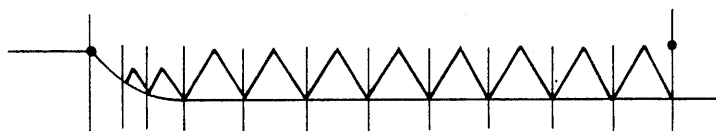
OUT = , IN = , FCE = .

Select, therefore, any among OUT, IN and FCE.

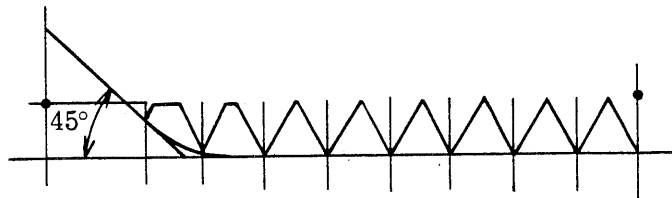
b. "CHAMFERING 60°(2), 45°(1), NO(0)?"

The shape of the chamfering can be set in terms of the chamfering angle. Specify "0" if no chamfering is required. Select "1" for 45° chamfering and "2" for 60° chamfering.

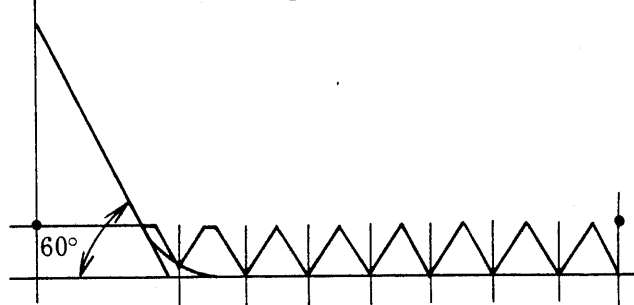
(0) is set : No chamfering.



(1) is set : 45° chamfering

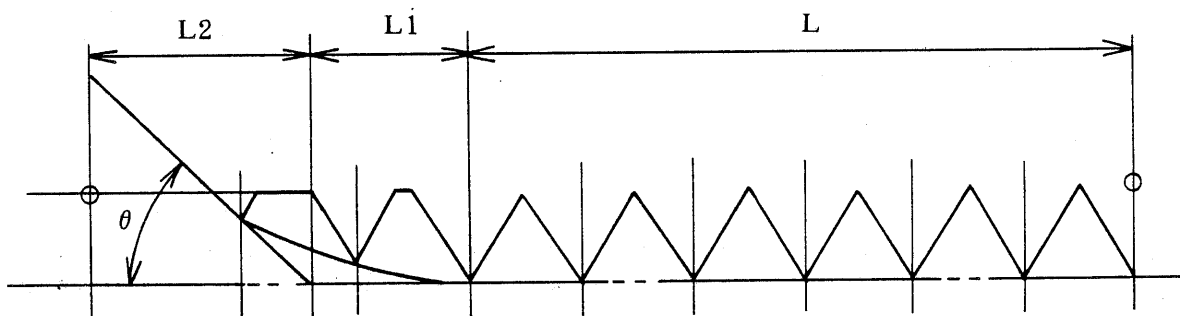


(2) is set : 60° chamfering



o Element determining amount of chamfering in THR:

Chamfering is effective starting at the position located at a distance equivalent to the value of the parameter **U36** from the final point.



L : Effective threads

L_1 : Incomplete threads at even pitches
(delay in follow-up)

Parameter **U36**

L_2 : Incomplete threads at even pitches
(amount of chamfering)

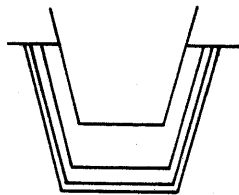
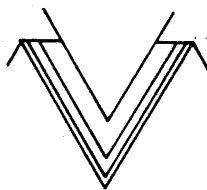
θ : Chamfering angle

c. "THREAD LEAD?"

Specify the lead of threads.

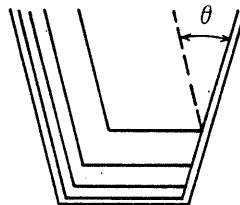
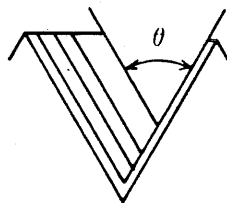
d. "ANGLE OF THREAD?"

Specify the cutting angle in threading. Set "0", if two-tool tip side cutting is to be specified.



Two-tool tip side cutting

Set "60" for millimeter thread or "55" for inch thread if single-tool tip side cutting is to be specified.



Single-tool tip side cutting

- o If the pitch is 3 or more, it is desirable to specify the thread angle.

e. "NUMBER OF ENTRANCE?"

Set the number of threads.

f. "THREAD HEIGHT (AUTO → MENU)?"

Set the height of threads (in mm).

Push the AUTO ? on the menu, the following are automatically determined:

f. HGT g. NUMBER h. V i. DEPTH

These values can be changed freely. They can also be set directly.

o THR mode thread height determining element:

Thread height = Thread pitch * parameter (see table below)

	Millimeter	Inch
THR OUT	U37	U39
THR IN	U38	U40
THR FCE	U37	U39

(Unit: % * 10⁻⁴)

Standard setting of U37 and U38: 6495 (thread angle: 60°)

Standard setting of U39 and U40: 6403 (thread angle: 55°)

* The standard setting are those recommended by JIS.

g. "NUMBER OF CUT PASSES?"

Specify the number of cuttings in threading.

o Element determining the number of cutting in THR mode:

The value automatically calculated by NC based on the lead and constants are entered.

h. "SURFACE SPEED?"

Set the surface speed in threading (in m/min.).

- o Element determining surface speed in THR mode: CUTTING
CONDITION picture
(THR : RV (m/min)) * (MATERIAL code (%))

i. "DEPTH OF FIRST CUT?"

Set the amount of first cut in threading (in mm).

- o Element determining amount of first cut in THR mode:
The value automatically calculated by NC based on the
lead and constants is entered.

j. "GEAR NO.?"

Set the spindle gear No. for threading.

If nothing or "0" has been set, the gear number is
determined automatically.

k. "TOOL NO.?"

Set the tool No. for threading.

"OFFSET NO.?"

Set the offset No. of the tool used for threading.

3-7-2 THR sequence data

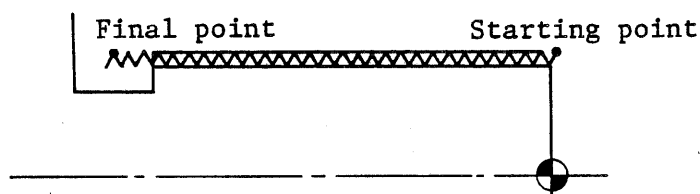
SEQ	SPT-X	SPT-Z	FPT-X	FPT-Z
*	a.	a.	b.	b.
*			c.	c.
			:	:
			:	:

a. "STARTING POINT-X?" "STARTING POINT-Z?"

Set the starting point in threading. The starting point-X sets the nominal diameter of the thread.

THR OUT

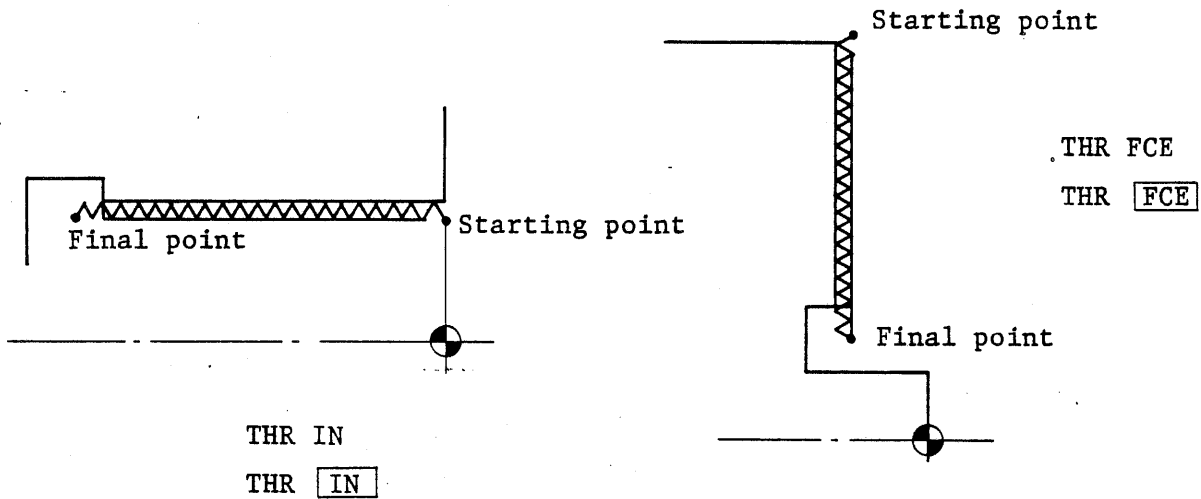
THR OUT



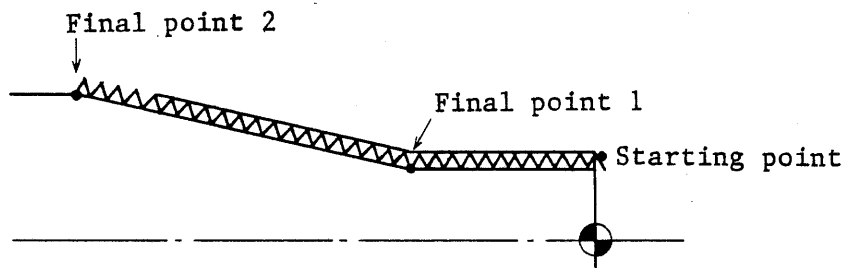
b. c. FINAL POINT-X?" "FINAL POINT-Z?"

Set the final point in threading.

The final point-X usually sets the nominal diameter of the thread. In the above figure, the final point is located near the center of the groove because incomplete threading is likely to occur near the final point.

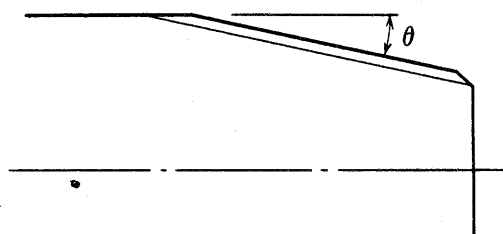


If a plural number of final points are specified as seen in c., continuous threading is performed. In continuous threading, only final points should be set.



Supplement

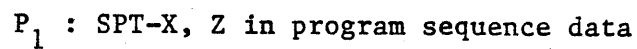
Taper threading command



The taper threading above is performed by entering the coordinate value of SPT-X and FPT-X. Machining part is selected according to the angle as follows:

$0 < \theta \leq 45^\circ$: Machining part is OUT or IN.

$45^\circ < \theta \leq 90^\circ$: Machining part is FCE.



P₂ : FPT-X, Z in program sequence data

L_0 : Threading acceleration distance

Threading acceleration distance is provided to prevent the pitch error due to follow-up delay in acceleration of the motors (servo-motors) driving the axes. The acceleration distance is calculated automatically by the NC based on the parameter A2 threading acceleration pitch error rate and parameter LP threading acceleration distance calculation constant.

To prevent an excessive acceleration distance, the maximum value is set in the parameter **U35**. If the calculated value exceeds the value set in the parameter **U35**, the alarm 532 "ACCELERATION DISTANCE EXCEED" is caused. Whether the threading acceleration distance is valid or invalid is determined by setting "0" or "1" in the parameter **U21**.

- 0 : Machining is executed even if the threading acceleration distance is insufficient.
- 1 : An alarm is caused if the threading acceleration distance is insufficient.

If an alarm is caused, perform the following:

1. Decrease threading rate (spindle r.p.m. x thread pitch).
2. Increase the parameter **A2** threading acceleration pitch error rate.
3. Increase the parameter **U35** acceleration distance clamp value.

L : Effective thread portion

L_1 : Incomplete thread portion due to follow-up delay


L_2 : Incomplete thread portion due to chamfering

Set this value in the parameter **U36**.

ψ : Chamfering angle

U34: Set the threading clearance in the parameter **U34**.

3-8 GRV : Groove Machining

This is used for groove machining. Select the  from the menu.

3-8-1 GRV process data

PNO MODE PART # NO. PITCH GRV WID FINISH RV FV FEED DEPTH GR RT FT
 * GRV a. b. c. d. e. f. g. h. i. j. k. l. m.

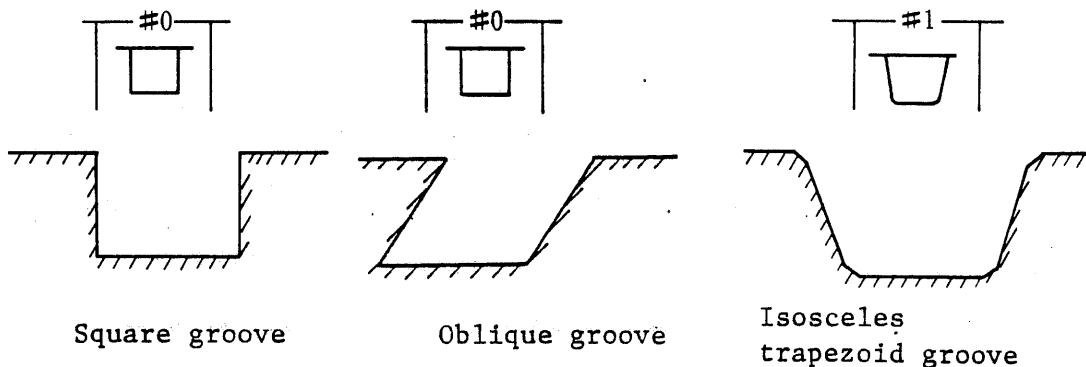
a. "MACHINING PART (MENU)?"

Select the range of machining from the menu. There is no difference between the open type and halfway type:

OUT = , IN = and FCE = .

Therefore, select any of OUT, IN and FCE.

b. "GRV: TRAPEZOID(1), STRAIGHT(0)?"



Select the type of the groove.

Set "0" for a square or oblique groove.

Set "1" for an isosceles trapezoid groove.

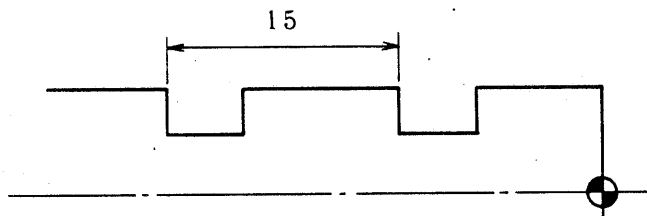
- o If nothing is set, it is regarded as setting "0".
- o If "0" is set as type of groove, only finish cutting is performed.
- o If "1" is set, both rough and finish cutting are performed.

c. "NUMBER OF GROOVES?"

Set the number of grooves at even intervals.

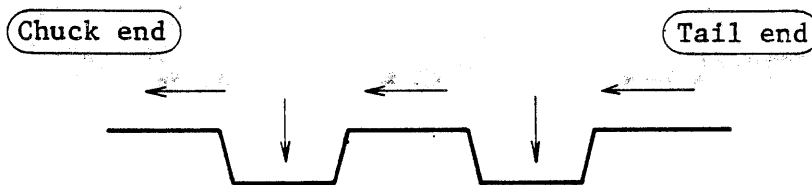
d. "GROOVING PITCH?"

If several grooves are going to be cut, set their pitch (in mm).

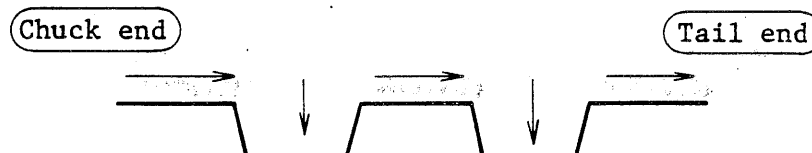


Number of grooves
(NO.) : 2
Pitch (PITCH) : 15

Note: The cutting direction varies according to the signs (plus and minus) of the pitch.



Pitch: Positive



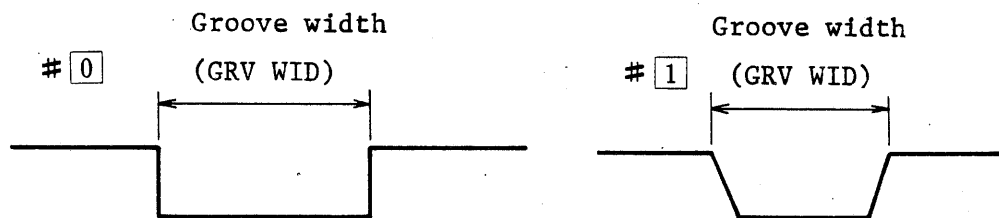
Pitch: Negative

If the pitch is positive, cutting is performed from the tail end to the chuck end.

If the pitch is negative, cutting is performed from the chuck end to the tail end.

e. "WIDTH OF GROOVE?"

Set the width of the groove (in mm).



In case "0" has been set as the type of groove, set the "GRV WID" as shown above.

If "1" has been set, set the width of the greater one. Always set this value.

f. "FINISH ALLOWANCE?"

Set the finishing allowance if "1" has been set as the type of groove.

If "0" has been set as the type of groove, setting of the finishing allowance is invalid.

g. "SURFACE SPEED FOR ROUGH CUT?"

Set the surface speed in rough cutting (in m/min.).

Select the AUTO ? from the menu, and the following will be determined automatically:

g. RV h. FV i. FEED j. DEPTH

- o Element determining roughing surface speed in GRV mode :
CUTTING CONDITION picture
(GRV : RV (m/min)) * (MATERIAL code (%))

These values can be changed freely. They can also be set directly.

- h. "SURFACE SPEED FOR FINISH CUT?"

Set the finishing surface speed for the groove type "1"
(in m/min). This can also be set automatically from the
menu.

- o Element determining surface speed in GRV mode: CUTTING
CONDITION Picture
(GRV: FV (m/min) * MATERIAL code (%))

- i. "FEEDRATE (/REV) FOR ROUGH CUT?"

Set the feedrate for rough cutting with the groove type
set at "1" and feedrate per revolution of the spindle for
groove type "0" (in mm/rev.).

- o Element determining feedrate per revolution in GRV mode:
rough cutting : CUTTING CONDITION picture
The value of (GRV: FEED (mm/rev.)) is entered.

In finish cutting for the groove type "1", feedrate is
determined based on the SRF code in the sequence data.

- j. "DEPTH OF CUT?"

Set the maximum amount of cut per path in rough cutting,
in terms of the radius (single side thickness) (in mm).

- o Element determining maximum amount of cut in GRV mode
rough cutting: CUTTING CONDITION picture

The value of (GRV : DEPTH (mm)) is entered. If the automatically determined value is to be used, proceed to the next step, using the INPUT key or cursor key.

- k. "GEAR NO.?"

Set the spindle gear No. for rough cutting.

If nothing or "0" has been set, the gear number is determined automatically.

- 1. "ROUGHING TOOL NO.?"

Select the groove type "1", and the roughing tool (RT) indication will appear. Set the tool No. for rough cutting.

If the groove type "0" has been selected, roughing tool (RT) cannot be entered.

"OFFSET NO.?"

Set the offset No. of the tool used for rough cutting.

- m. "FINISHING TOOL NO.?"

In case of the groove type "1", set the tool No. for finish cutting after setting of roughing tool (RT).

If the groove type "0" has been selected, set the tool No. for groove machining.

"OFFSET NO.?"

Set the offset No. of the tool used for finish cutting.

3-8-2 GRV sequence data

The sequence data in GRV mode are somewhat different between the groove type #0 and #1.

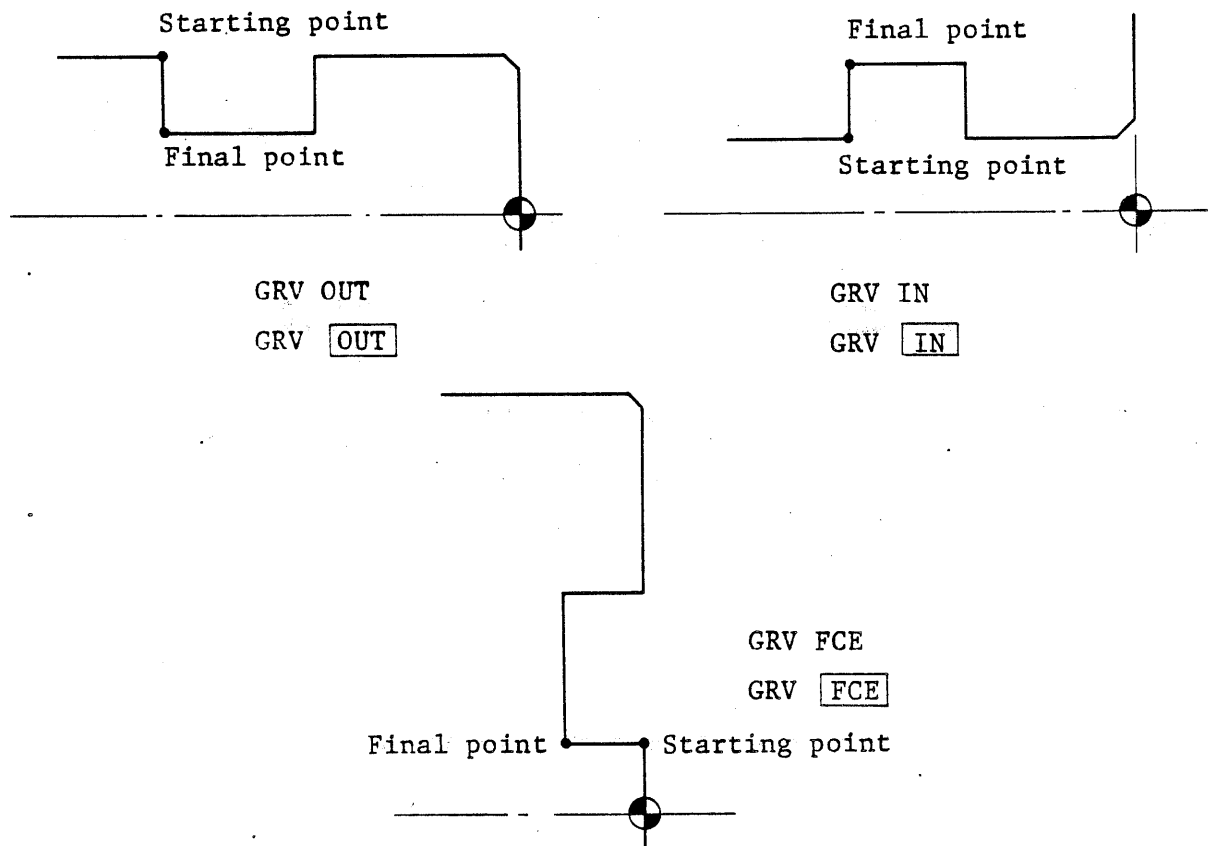
(i) Groove type #0:

SEQ	S-CNR	SPT-X	SPT-Z	FPT-X	FPT-Z	F-CNR	RADIUS	SRF	\$	M
*		a.	a.	b.	b.		c.			d.

Although S-CNR, F-CNR, SRF and \$ are displayed for the groove type #0, these data are invalid even if entered.

a. "STARTING POINT-X?" "STARTING POINT-Z?"

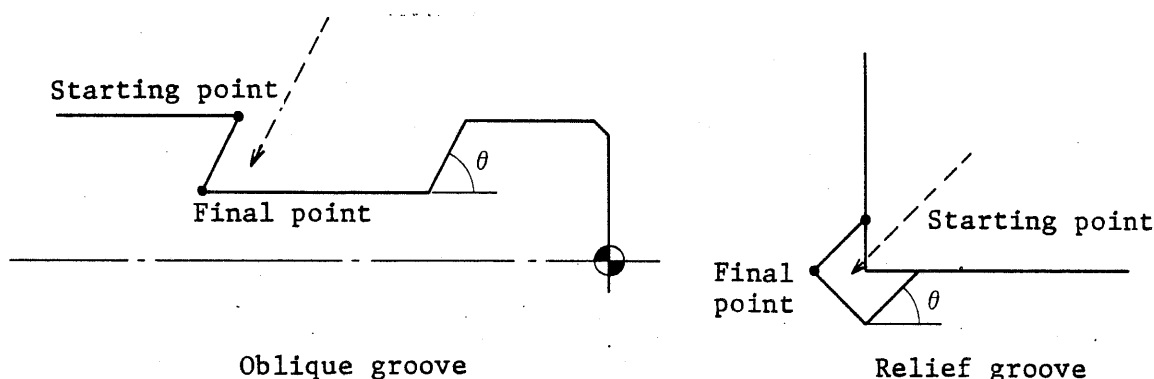
Set the starting point of cutting for the groove type #0. The starting point should be determined as shown in the figure below:



b. "FINAL POINT-X?" "FINAL POINT-Z?"

Set the final point of cutting for the groove type #0. (For definition of the final point, see the figure above.)

A relief groove or oblique groove can be cut by varying the values of SPT-Z and FPT-Z.



If any one of the SPT-X, SPT-Z, FPT-X and FPT-Z is unknown in machining of an oblique groove or relief groove; enter $\boxed{?}$ on the menu for such unknown value and set the oblique angle or relief angle in the c. RADIUS, and the unknown value of the starting point or final point can be found. (If the $\boxed{?}$ is entered, a value can be put in the column for RADIUS.)

$\boxed{?}$ can be entered in only one of the SPT-X, SPT-Z, FPT-X, or FPT-Z. If more than two $\boxed{?}$ are set, no calculations will be possible.

(See 6-4-2 Intersect point computation for taper, GRV sequence.)

c. "TAPER ANGLE?"

If the starting point or final point is unknown in cutting of an oblique groove or relief groove, set the oblique angle or relief angle.

Be careful in affixing a sign because the calculated value will be different if a sign is entered incorrectly. (Negative sign for the above figures)

d. "M CODE (MENU)?"

An M code can be set with the menu or ten keys. This is an auxiliary function. The M function becomes effective when finish cutting begins. (Machining for the groove type #0 is regarded as finish cutting because the tool is set at FT.)

For M codes, refer to M (M function) of process data.

Note : "SRF" in sequence data is invalid, but in LAYOUT picture it is displayed as finish process.

(ii) Groove type #1

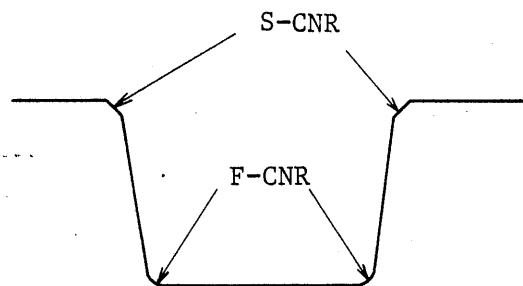
SEQ	S-CNR	SPT-X	SPT-Z	FPT-X	FPT-Z	F-CNR	RADIUS	SRF	\$	M
*	a.	b.	b.	c.	c.	d.	e.	f.	g.	h.

Different from the groove type #0, S-CNR, F-CNR and SRF for the groove type #1 can be entered as sequence data. If "4" is set in grinding allowance feedrate per minute can be specified. However, input of 1, 2, 3, 5, 6 and 7 is invalid.

a. "STARTING CORNER?"

d. "FINAL CORNER?"

An isosceles trapezoid groove of the type #1 can be chamfered or rounded at the corner.

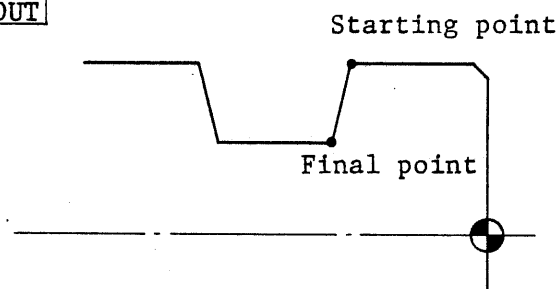


b. "STARTING POINT-X?" "STARTING POINT-Z?"

Set the starting point of cutting for the groove type #1. Note that the positions of the starting point and final point are reversed as compared with those for the groove type #0.

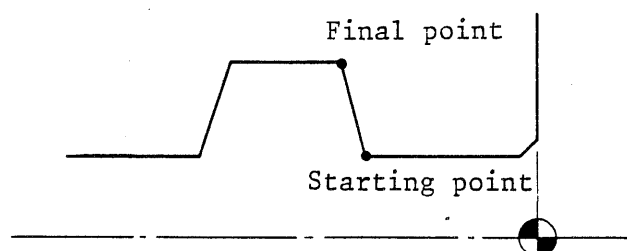
GRV OUT

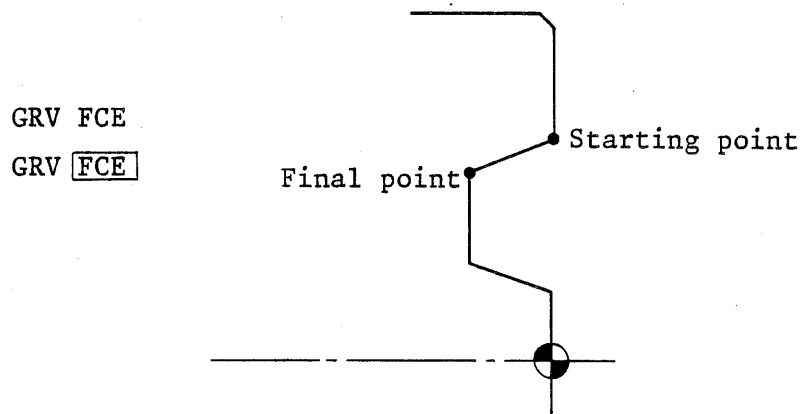
GRV ☐ OUT



GRV IN

GRV ☐ IN



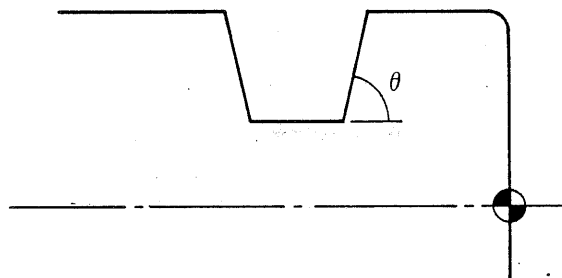


- c. "FINAL POINT-X?" "FINAL POINT-Z?"

Set the final point of cutting for the groove type #1. (See the above figure.)

- e. "TAPER ANGLE?"

If any of the values of the starting point and final point is unknown, such value can be automatically calculated by specifying the taper angle.



- f. "FIN SURFACE ROUGHNESS (MENU)?"

The feedrate in finish machining is automatically calculated and set if the finish surface roughness (SRF) code selected from the finish codes on the menu is entered.

g. "NECKING OR FR/MIN (MENU)?"

If "4" is set here after setting F.SRF, feed per minute is set. (See description of the grinding allowance in the sequence data on BAR.)

Setting of 1, 2, 3, 5, 6 and 7 of the grinding allowance code is invalid.

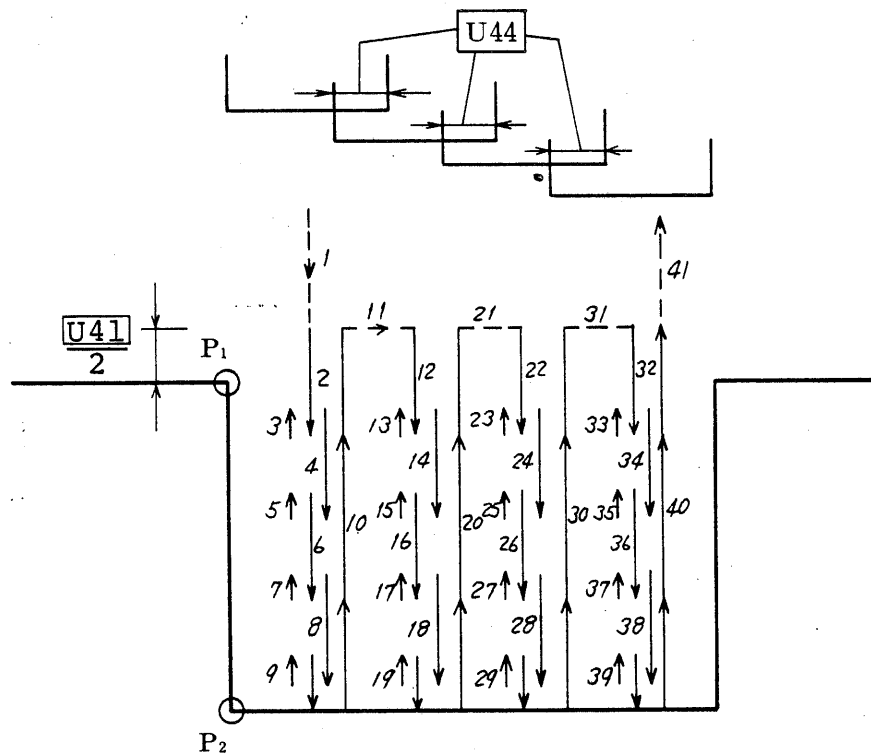
h. "M CODE (MENU)?"

An M code can be set with the menu or ten keys. This is an auxiliary function. The M code thus set becomes valid when finish machining begins.

For M codes, see the description of M codes in process data.

3-8-3 Tool path in GRV OUT and GRV OUT

For #0



P_1 : SPT-X, Z set by program sequence data

P_2 : FPT-X, Z set by program sequence data

U41: Grooving clearance (X-axis) set by parameter U41 .

U44: The amount of grooving overlap is set by the parameter U 44 .

Notes :

(1) The tool paths 2 through 10 are the same. Similarly, the

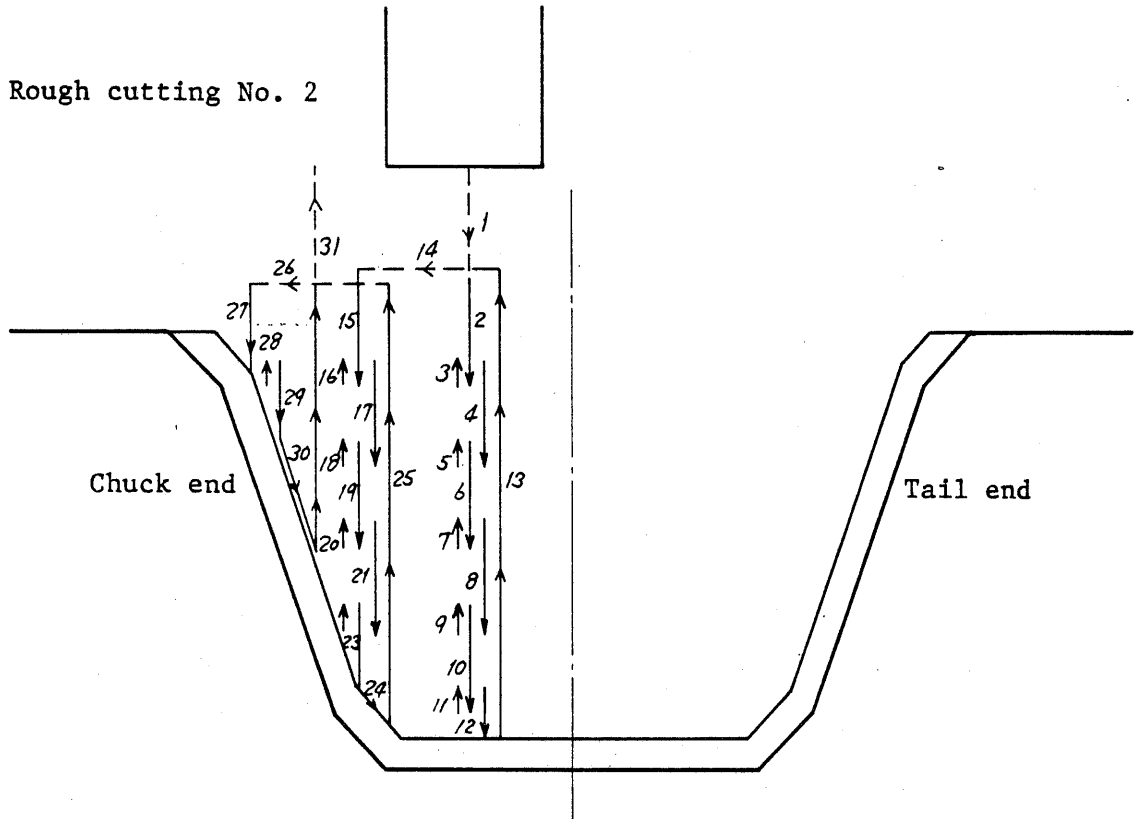
tool paths 12 through 20, 22 through 30 and 32 through 40 are the same.

- (2) The tool paths 3, 5, 7, 9, 13, 15, 17, 19, 23, 25, 27, 29, 33, 35, 37 and 39 are the amounts of return in pecking. They are set by the parameter U43. Pecking operation is conducted, irrespective of the type of groove.
- (3) The tool paths 10, 20, 30 and 40 in the figure is the return from the bottom of the groove at the rate set by the parameter U45.

3-8-5 Tool path in GRV OUT and GRV OUT

For #1 .

Rough cutting No. 2



P_1 : SPT-X, Z set by program sequence data

P_2 : FPT-X, Z set by program sequence data

F : Finishing allowance set by program sequence data

U41: Grooving clearance (X-axis) set by parameter U41

U44: The amount of grooving overlap is set by the parameter

U44

NOTES:

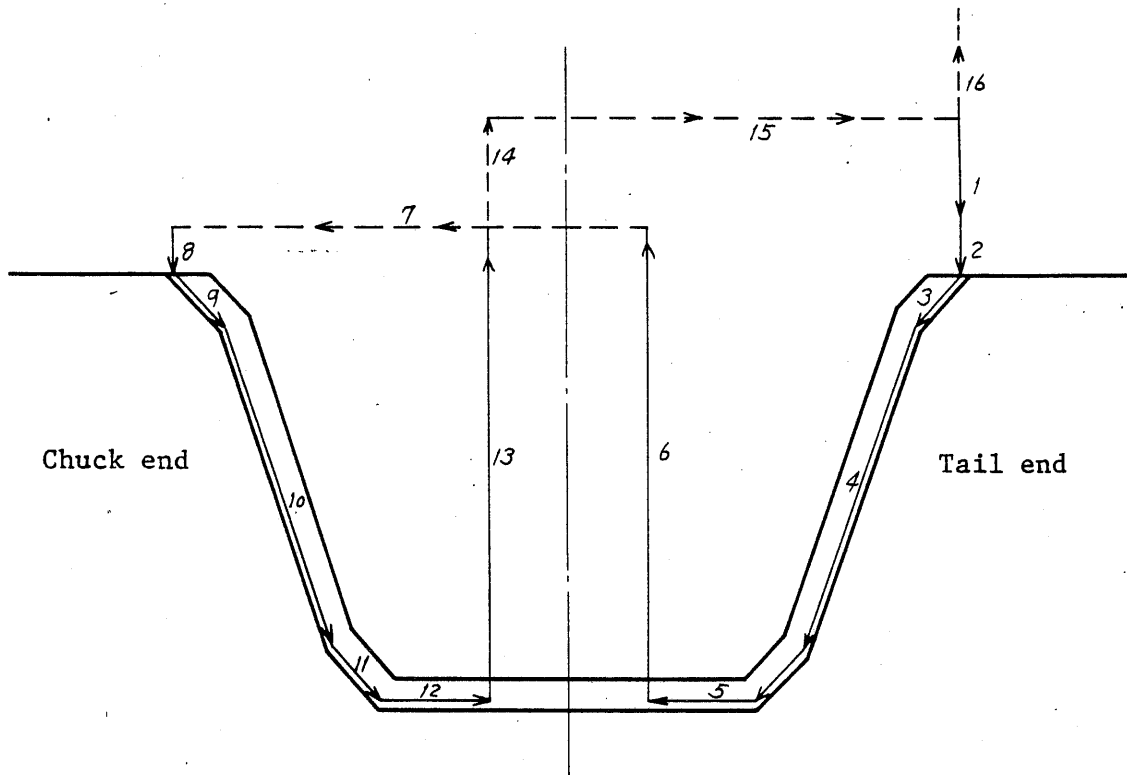
- (1) The tool paths 2 through 13 in rough cutting No. 1 are the same. Similarly, the tool paths 15 through 26 are the same. The tool paths 2 through 13 in rough cutting No. 2 are the same.

- (2) In the above figure, 3, 5, 7, 9, 11, 16, 18, 20, 22, 24, 29, 31, 33, 35 and 40 are the return amounts of pecking in grooving. They are set in the parameter U43. Pecking operation is performed, irrespective of the type of groove.
- (3) The tool paths 13, 26, 37 and 43 are the return from the bottom of the groove at the rate set by the parameter U45.
- (4) The dwell is actuated and the tool is stopped at the bottom of the groove for a period of time equivalent to the spindle r.p.m. set by the parameter P18.
- (5) Machining cycle in rough cutting is more than 3.

3-8-6 Tool path in GRV OUT and GRV OUT

For #1

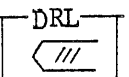
Finish cutting



Note: The tool path in finish cutting is from 1 through 16 in the above figure.

3-9 DRL : Drilling

It is used to drill a hole in the center of a workpiece.

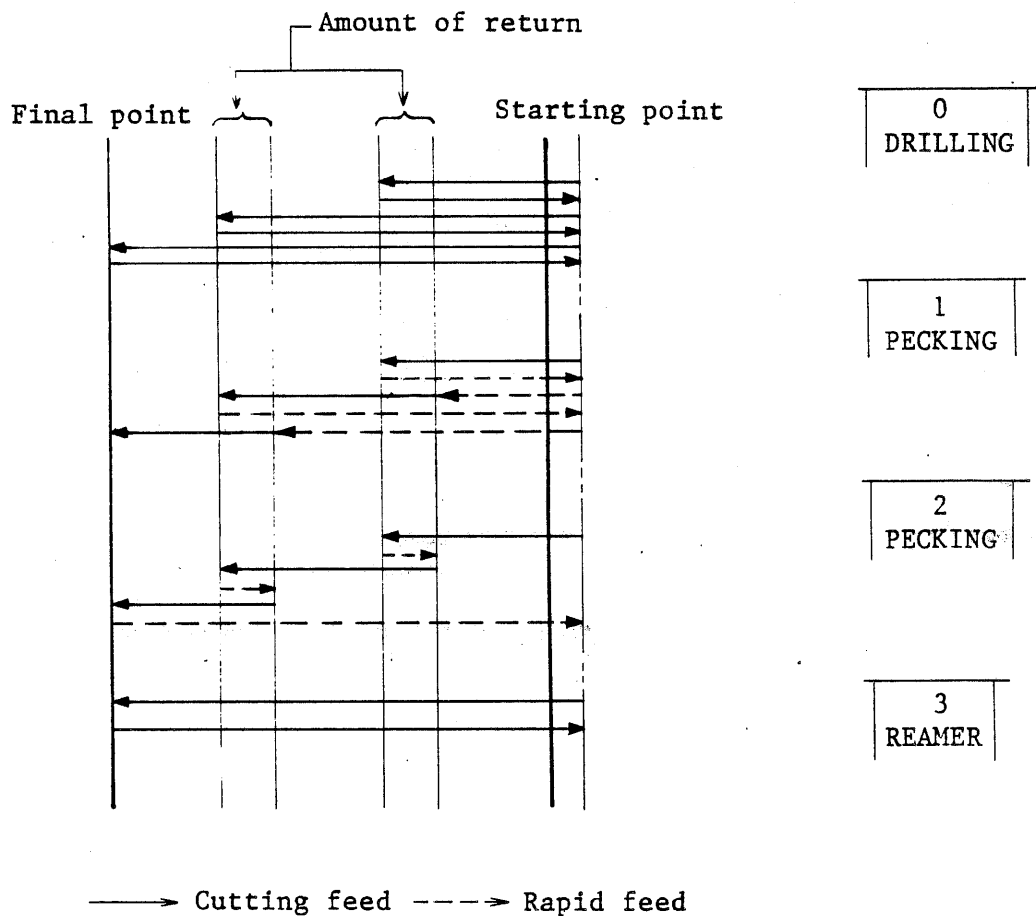
Select the  on the menu.

3-9-1 DRL process data

PNO	MODE	#	DRL	DIA	DEP-1	DEP-2	DEP-3	V	FEED	GR	T
*	DRL	a.	b.	c.	d.	e.	f.	g.	h.	i.	

a. "DRILLING TYPE (MENU)?"

Select a pattern of drilling from the menu.



b. "DRILL DIAMETER?"

Set the diameter of the drill used for drilling.

Register the diameter of the drill actually used in the column of ACT- ϕ on the TOOL DATA picture.

c. "DEPTH OF FIRST CUT?"

Set the amount of first cut in drilling (in mm). Push the AUTO ?, and the following will be determined automatically.

c. DEP-1 d. DEP-2 e. DEP-3 f. V g. FEED

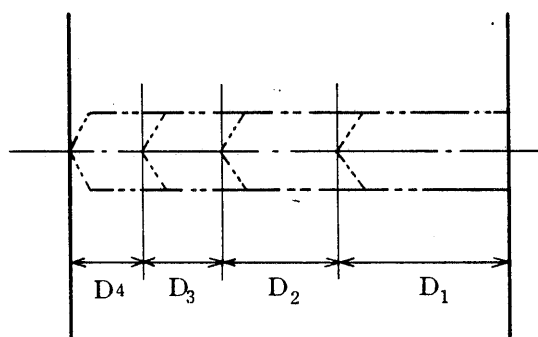
o Element determining first cut in DRL mode:

$$(\text{Drill diameter (mm)}) * \left(\frac{\text{Parameter } \boxed{\text{U51}}(\%)}{100} \right)$$

The automatically determined value can be changed freely.
Direct setting of the value is also possible.

d. "DECREMENTAL DEPTH OF CUT?"

Set the decrement in depth of each drilling operation (in mm).



$$\begin{aligned} D_1 - D_2 &= \alpha \\ D_2 - D_3 &= \alpha \\ D_4 - D_3 &= \alpha \\ \alpha &: \text{Decrement} \end{aligned}$$

- o Element determining decrement in depth in DRL mode:

The depth of cut decreases in each drilling operation by the amount equivalent to (Parameter U49 (0.001mm)).

- e. "MINIMUM DEPTH OF CUT?"

Set the minimum depth of cut in drilling (in mm). As the number of cutting operations increases, the amount of depth of cut decreases by the amount equivalent to that described in d. above. When the minimum amount of cut is reached, the amount of cut decreases no more and the remaining portion is cut without any decrement.

- o Element determining minimum amount of cut in DRL mode:

The amount of cut is clamped by (Parameter U53 (0.001 mm)).

- f. "SURFACE SPEED?"

Set the surface speed in drilling (in m/min.).

- o Element determining surface speed in DRL mode:
CUTTING CONDITION picture

$(\text{DRL} : R V \text{ (m/min.)}) * (\text{MATERIAL code } (\%))$

If the automatically determined value is to be used, proceed to the next step, using the INPUT key or cursor key.

- g. "FEEDRATE (/REV)?"

Set the feedrate per revolution of the spindle in drilling (in mm/rev.).

- o Element determining feedrate per revolution in DRL mode:
CUTTING CONDITION picture

The value of (DRL : FEED (mm/rev.)) is entered.

- h. "GEAR NO.?"

Select the spindle gear No. in drilling. The gear number is automatically determined if nothing or "0" has been set.

- i. "TOOL NO.?"

Set the tool number for drilling.

"OFFSET NO.?"

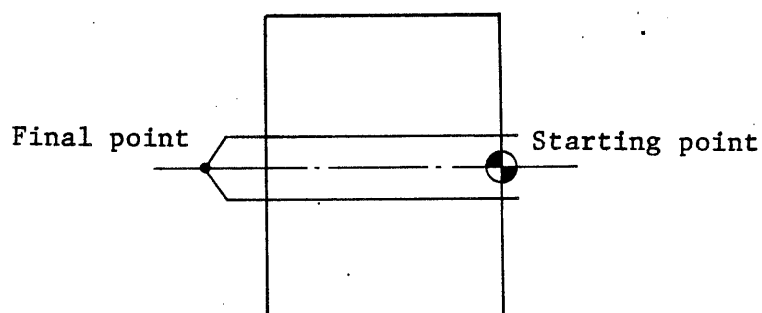
Set the offset number of the tool used for drilling.

3-9-2 DRL sequence data

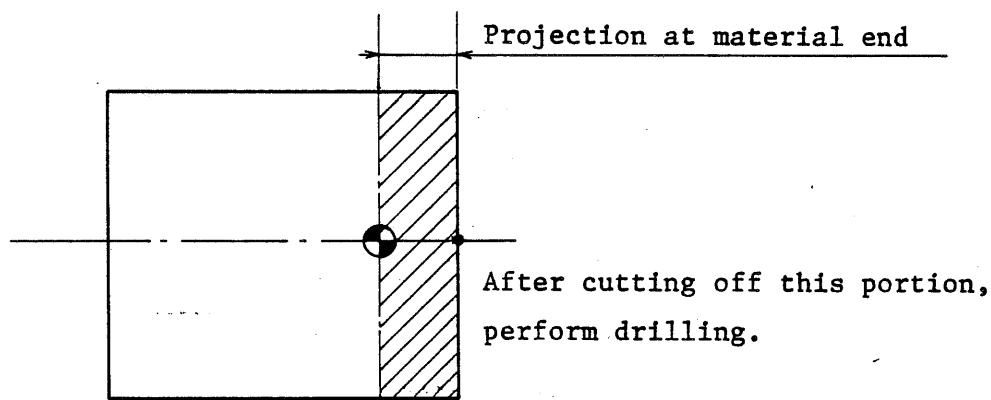
SEQ	SPT-X	FPT-Z	M
*	a.	b.	c.

- a. "STARTING POINT-Z?"

Enter the starting point in drilling.



A positive value shall be entered for SPT-Z. If a projection exists as in the figure for example, drilling is performed after cutting off the material end.



b. "FINAL POINT-Z?"

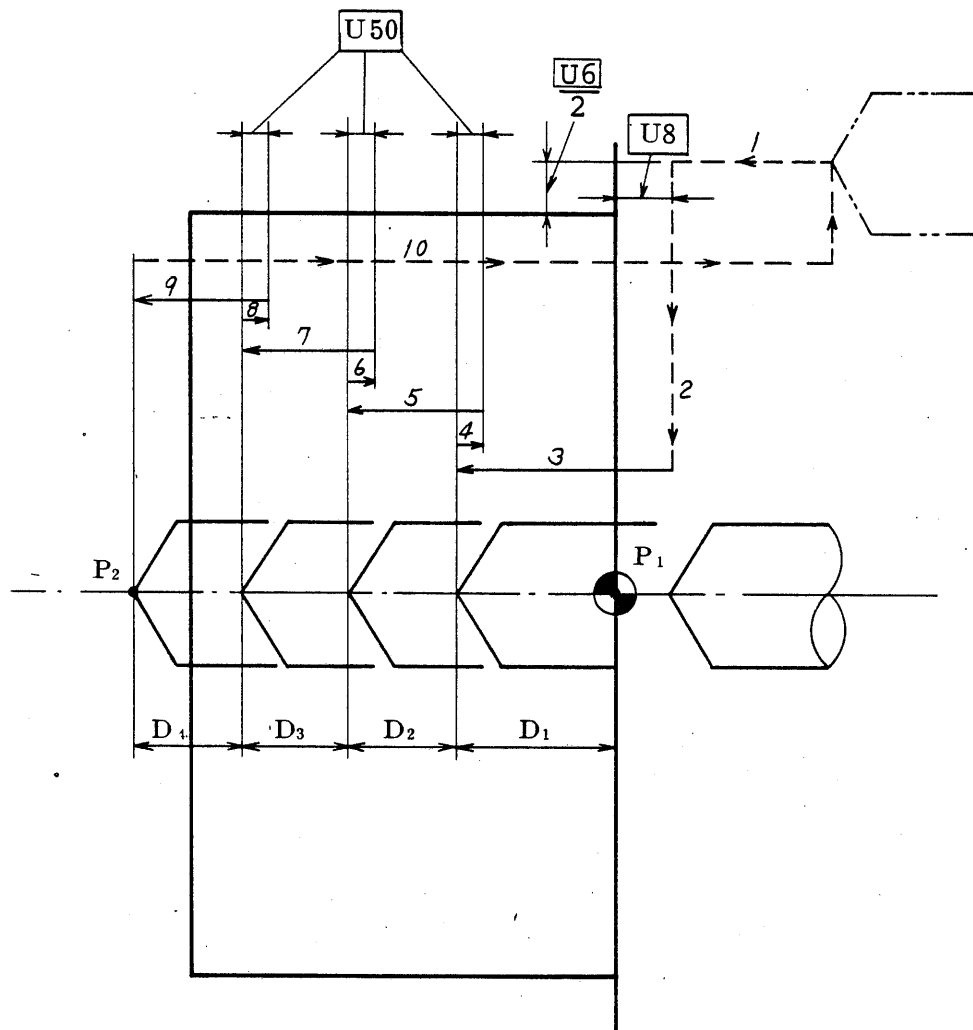
Enter the final point in drilling.

Drilling is completed when the drill tip has reached the final point. Depending on the angle of the drill tip, some portion may be left undrilled when a through hole is to be bored. Consider this in setting the final point. (See the figure above.)

c. "M CODE (MENU)?"

An M code can be entered with the menu or ten keys. This is an auxiliary function. The M function becomes effective when finish cutting begins.

3-9-3 Tool path for DRL #2



-----> Rapid feed
 ———> Cutting feed

P_1 : SPT-Z in program sequence data

P_2 : FPT-Z in program sequence data

U6 : Safety contour clearance (O.D.) set by parameter

U6

U8 : Safety contour clearance (face) set by parameter

U8

U50: Relief amount of drill set by parameter U50

D_1 : Amount of first cut with drill

If it is to be determined automatically, the value of the parameter U51 is used.

$$D_1 = \text{Drill diameter} * \boxed{\text{U51}} / 100$$

D_2 through D_4 : Amounts of drill cut

Suppose that the amount of first cut is d_1 and that the amount of i -th cut is d_i :

$$d_i = d_1 - \boxed{\text{U49}} * (i - 1) \text{ (When } d_i \geq b \text{)}$$

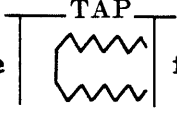
The above equation remains valid until the b (drill cut clamp value (minimum cut)) is reached. This clamp value is set in the parameter $\boxed{\text{U53}}$.

$$d_i = b \text{ (when } d_i < b \text{)}$$

Note : The tool paths 3 through 10 are the same. They are located on the centerline of the spindle.

3-10 TAP : Tapping

This function is used to tap a hole in the center of a workpiece.

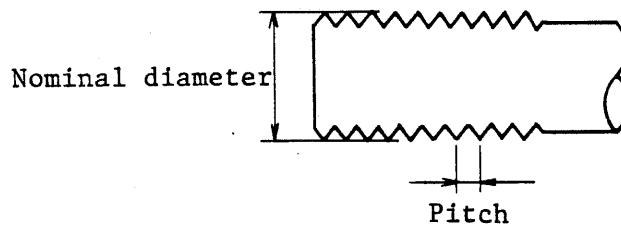
Select the  from the menu.

3-10-1 TAP process data

PNO	MODE	NOM-D	PITCH	V	GR	T
*	TAP	a.	b.	c.	d.	e.

a. "NOMINAL SIZE OF TAP?"

Set the nominal diameter of the tap. (See the figure below.)



b. "PITCH?"

Set the pitch of the tap. (See the figure above.)

c. "SURFACE SPEED?"

Set the surface speed in tapping (in m/min.).

If the **AUTO ?** on the menu is pushed, the surface speed will be determined automatically.

- o Element determining surface speed in TAP mode:
 (TAP: RV (m/min.)) * (MATERIAL code (%))

The automatically determined value can be changed freely.
 It can also be set directly.

d. "GEAR NO.?"

Select the spindle gear No. in tapping. Nothing or "0" has been set, the gear number is determined automatically.

e. "TOOL NO.?"

Set the tool No. for tapping.

"OFFSET NO.?"

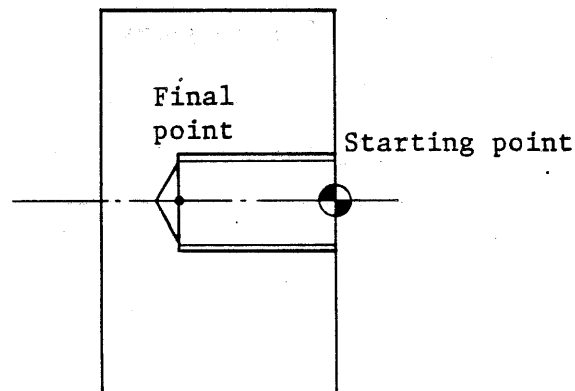
Set the offset No. used for tapping.

3-10-2 TAP sequence data

SEQ	SPT-Z	FPT-Z	M
*	a.	b.	c.

a. "STARTING POINT-Z?"

Enter the starting point of tapping.



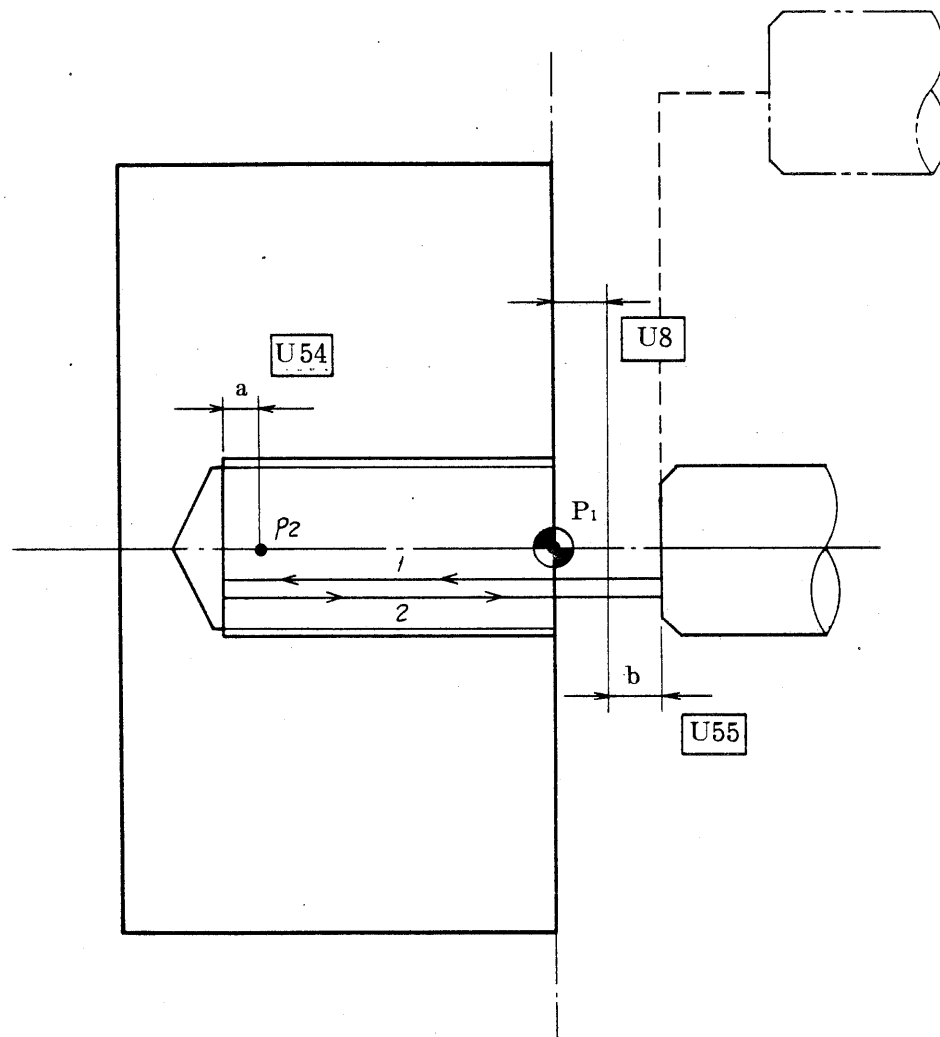
b. "FINAL POINT-Z?"

Enter the final point of tapping.

c. "M CODE (MENU)?"

The M code can be entered with the menu or ten keys auxiliarily. The M function becomes effective when finish machining begins.

3-10-3 Tool path in TAP



- Rapid feed
- Cutting feed

P_1 : SPT-Z in program sequence data
 P_2 : FPT-Z in program sequence data
 $U6$: Safety contour clearance (O.D.) set by parameter $U6$
 $U8$: Safety contour clearance (face) set by parameter $U8$
 a, b : Used for offset of the tap path with the incomplete threads and elongation of the tap taken into account.
 Offset of incomplete threads:

$$a = \text{Pitch (tap pitch)} * \frac{\boxed{U54}}{10}$$

Offset for elongation of tap

$$b = \text{Pitch} * \frac{\boxed{U55}}{10}$$

Note : Tool paths 1 and 2 are the same. They are located on the centerline of the spindle.

3-11 MNP (Manual Program Mode) : Manual Program Machining

The manual program mode supplements the functions in the automatically developed machining mode.

Push the

MANUAL PROGRAM

 on the menu.

3-11-1 MNP process data

PNO	MODE	CHANGE-PT	GR	T
*	MNP	a.	b.	c.

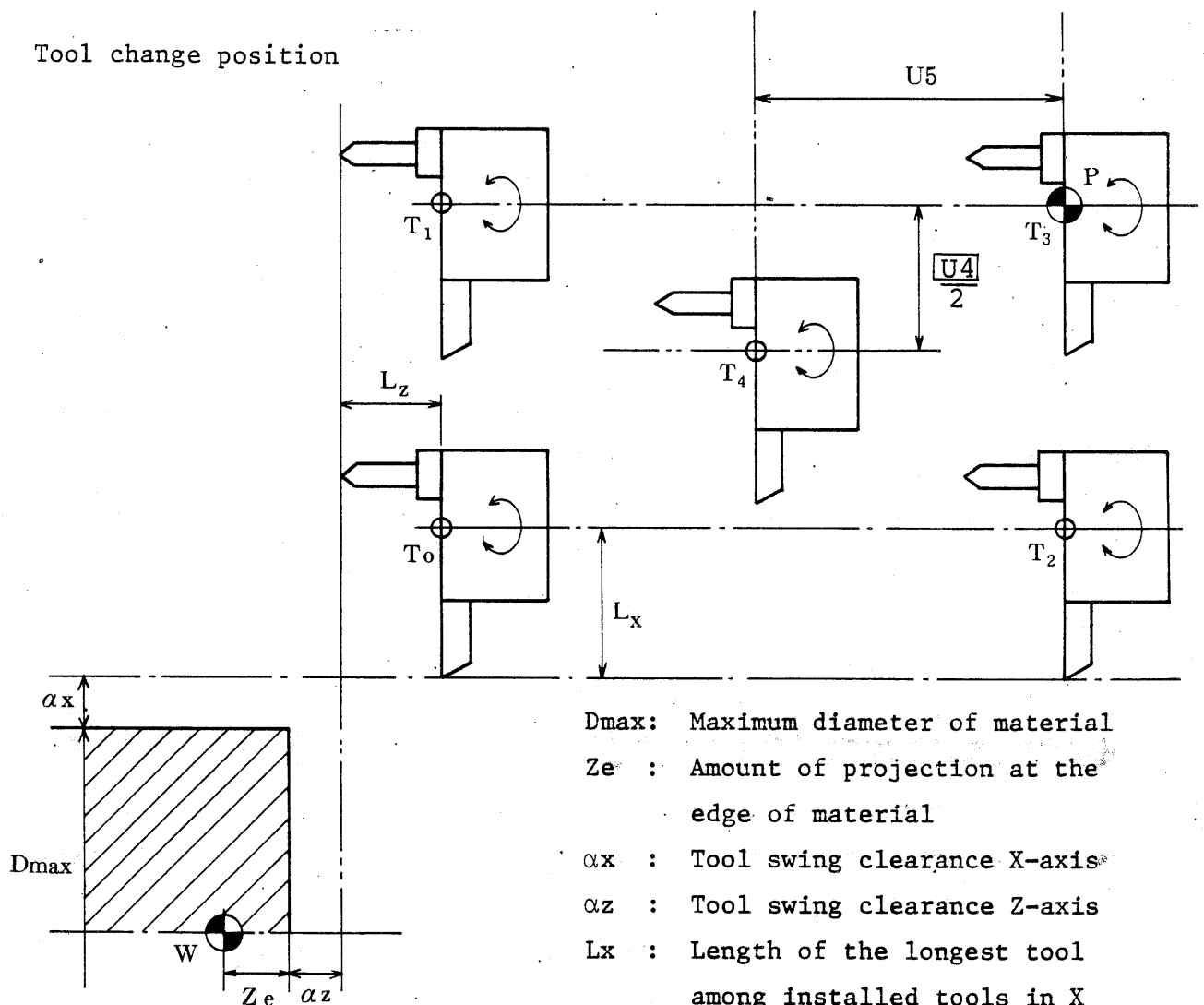
a. "EXCHANGE PT OF TOOL Y (1), N(0)?"

If it is desired to return the tool to the tool change position in the beginning of the manual program mode, set "1". Otherwise, set "0".

If "1" has been set for CHANGE-PT, the tool moves to the tool change position before executing SEQ. If the tool the same as for previous process is to be used in MNP, however, it moves to the safety point.

If "0" has been set for CHANGE-PT, the tool is replaced at the position where machining has been completed in the previous process. Be careful because the tool will interfere with a workpiece if a different mode precedes the manual program mode or if a program containing the manual program mode is connected to any other program.

The tool change position is determined by entering any figure of 0 through 4 in the user parameter P2.



D_{max} : Maximum diameter of material

Z_e : Amount of projection at the edge of material

α_x : Tool swing clearance X-axis

α_z : Tool swing clearance Z-axis

L_x : Length of the longest tool among installed tools in X direction

L_z : Length of the longest tool among installed tools in Z direction

W : Program zero point

P : Machine zero point

Setting of parameter <u>P2</u>	Position in above figure	X	Z
0	T_0	Clearance posi- tion αx	Clearance posi- tion αz
1	T_1	X-axis machine zero point	Clearance posi- tion αz
2	T_2	Clearance posi- tion αx	Z-axis machine zero point
3	T_3	X-axis machine zero point	Z-axis machine zero point
4	T_4	Fixed point	Fixed point

If "0" has been set, the tool swings in the following position:

X-axis : Position equivalent to the maximum O.D. + tool swing clearance (in the X direction) set by the user parameter U1

Z-axis : Position equivalent to Program zero point + amount of projection at material edge + tool swing clearance (in Z direction) set by the user parameter U2.

The tool sings in the following position if "1" has been set:

X-axis : Machine zero point position of X-axis

Z-axis : Program zero point + Projection at material edge + Tool swing clearance (parameter U2)

If "2" has been set, the tool swings in the following position:

X-axis : Maximum O.D. of material + Tool swing clearance
(set by parameter U1)

Z-axis : Machine zero point position of Z-axis

If "3" has been set, the tool swings in the following position:

X-axis : Machine zero point position of X-axis

Z-axis : Machine zero point position of X-axis

If "4" has been set, the tool swings in the following position:

X-axis : Fixed point position determined by user
parameter U4

Z-axis : Fixed point position determined by user
parameter U5

The fixed point can be set at any desired position. Set the distance from the machine zero point in the user parameters U4 (X direction) and U5 (Z direction).

The tool swings in any of the above five positions. The tool swing clearance described above is incorporated into the tool tip position of the longest tool among tools installed. (Longest tool in the X direction and longest tool in the Z direction).

Therefore, it is necessary to perform tool set and Z offset correctly after installation of tools.

b. "GEAR NO.?"

Set the spindle gear No. used for single process machining. If nothing is specified, the alarm 501 "ILLEGAL PROCESS DATA" is caused.

c. "TOOL NO.?"

Set the tool No. used for single process machining. If nothing is specified, the alarm 502 "ILLEGAL SEQUENCE DATA" is caused.

3-11-2 MNP sequence data

SEQ	G	FPT-X	FPT-Z	RADIUS	RPM	FEED RATE	M	OFS
*	a.	b.	b.	c.	d.	e.	f.	g.

a. "G CODE (MENU)?"

The following six can be selected from the menu as the interpolation preparation commands:

G00 : Positioning (rapid feed)
 G01 : Linear interpolation (cutting feed)
 G02 : Arc interpolation (clockwise : CW)
 G03 : Arc interpolation (counterclockwise : CCW)
 G04 : Dwell
 G32 : Threading interpolation

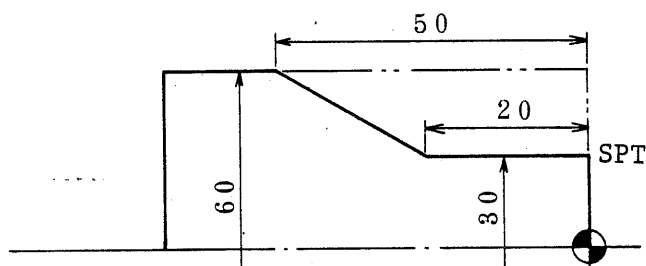
Of these six G codes, G00, G01, G02 and G03 are modal data (continuous data). G04 is non-modal data (discontinuous data).

b. "FINAL POINT-X?" "FINAL POINT-Z?"

Enter the values of the coordinates X and Z at the final point of the move command.

In case of dwell interpolation G04, enter the dwelling time in the FPT-X position.

Move commands are classified into absolute commands and incremental commands.



When absolute command is given.

When incremental command is given.

	FPT-X	FPT-Z		FPT-X	FPT-Z
	
SP	30	0	SP	0 ◀	0 ◀
	30	-20		0 ◀	-20 ◀
	60	-50		30 ◀	-30 ◀

Comparison of absolute command with incremental command with SP (in the above figure) used as starting point

As seen in the comparison above, a value with respect to the program zero point must be entered in case of the absolute command. That is, the coordinates in which the program zero point is regarded as zero are adopted.

In case of the incremental command, the amount by which shifting is to be executed must be entered from time to time. That is, the position to which shifting has been

accomplished must be used as the zero for the coordinates.

The sign of the command value is positive (negative) if the position is above the zero (below the zero) in case of the X direction. In case of the Z direction, the command value is positive (negative) if the position is at the right (left) of the zero point.

Ordinarily, the absolute command is used. If the incremental command is to be used, enter it with the INCRMENT on the menu inverted.

Display is made in green for the absolute command and in yellow for the incremental command. Figures are indicated with "◀" as seen in the above comparison.

Note : In sequence No. 1, it is impossible to enter data by incremental command. If this is done, the alarm 325 (525) "G90 IN IST BLK OF MNL PROG" is caused.

c. "CIRCLE RADIUS?"

Enter the radius of the arc used for arc interpolation in G02 and G03.

d. "S CODE (SURFACE SPEED V → MENU)?"

Enter the revolution of the spindle in terms of r.p.m. or surface speed.

- o R.P.M. command: Set the r.p.m.

If the r.p.m. command has been given, the constant r.p.m. control is executed. Display is made as "S□□□□".

- o Surface speed command: Set the surface speed in m/min. with the SURF SPD V on the menu inverted.

If the surface speed command has been entered, the constant surface speed control is executed. Display is made as "V □□□□".

e. "FEEDRATE (/REV) (/MIN→MENU)?"

Feedrate of X- and Z-axes are specified in terms of feedrate per revolution or per minute.

- o Feedrate per revolution : It means the synchronous feeding expressed in feedrate per revolution of the spindle. Enter the figure directly, and feedrate per minute will be effective. Display is made as "REV □□.□□□".
- o Feedrate per minute: It means asynchronous feeding expressed in feedrate per minute. Enter the figure with the FEEDRATE/MIN on the menu inverted. Display is made as "MIN □□□□ . □□□".

f. "M CODE (MENU)?"

An M code can be auxiliarily entered with the menu or ten keys.

- o The M function specified here becomes effective in the next sequence. If the next sequence is a program involving a feedrate per minute, the direction of the spindle rotation must be specified by M03.(forward) or M04 (reverse).

If the direction of the rotation has been set in the TOOL DATA, however, it need not be specified by M code.

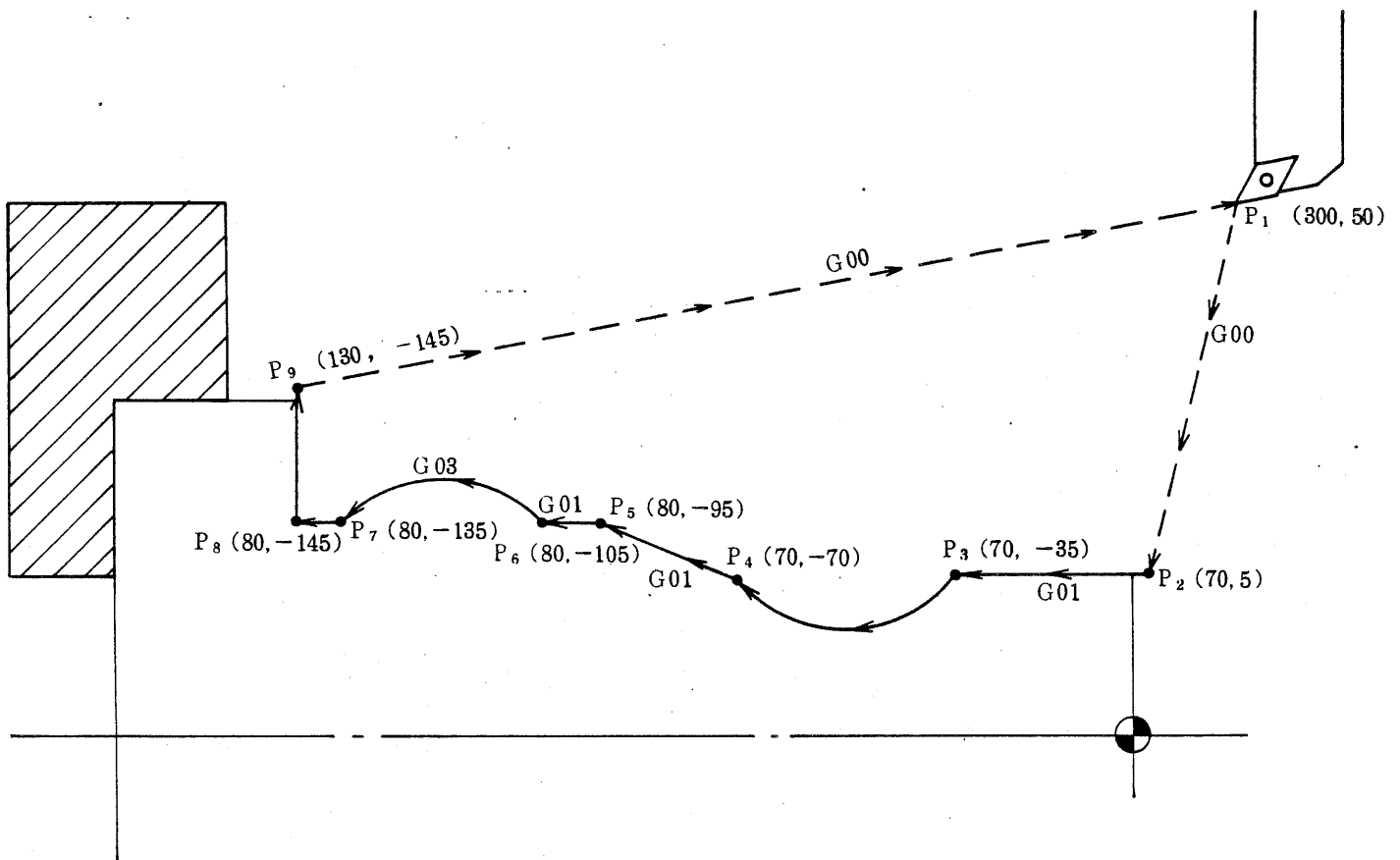
- o Do not specify M if threading interpolation in G32 has been set.

g. "OFFSET NO.?"

Specify the offset No. for the tool used in single process machining. Offset No. 1 and 2 are respectively corresponding to OFFSET-1 and OFFSET-2 on the TOOL SET picture. If nothing is specified, offset is not executed.

OFS "0" means to cancel offset.

3-11-3 Tool path in MNP



P₁ : Tool swing position
 P₂ : Approach point
 P₃ : Final point of line
 P₄ : Final point of arc
 P₅ : Final point of taper

P₆ : Final point of line
 P₇ : Final point of arc
 P₈ : Final point of line
 P₉ : Final point of line
 (including relief amount)

Programme

PNO	Mode	CHANGE-PT					GR	T
1	MNP	1					2	1
SEQ	G	FPT-X	FPT-Z	RADIUS	RPM	FEEDRATE	M	OFS
1	0	300	50		V130		4	1
2	0	70	5					
3	1	70	-35			REV 0.35		
4	2	70	-70	30				
5	1	80	-95					
6	1	80	-105					
7	3	80	-135	30				
8	1	80	-145					
9	1	130	-145					
10	0	300	50					

- (1) G00 : A menu to select the positioning mode in manual program mode.
Positioning is performed in rapid feed.
- (2) G01 : A menu to select the interpolation mode in manual program mode.
Positioning is performed in cutting feed.
- (3) G02 : A menu to select the arc interpolation (clockwise) mode in manual program mode. Positioning is performed in cutting feed.
- (4) G03 : A menu to select the arc interpolation (counterclockwise) mode in manual program mode. Positioning is performed in cutting feed.
- (5) G04 : Dwell
- (6) G32 : Threading interpolation in manual program mode.

Note : Shape display by FIGURE CHECK is not performed in MNP.

3-12 M (M Function) : M Code

The M function, which is an auxiliary function, is set.

Select the M-CODE from the menu.

3-12-1 M process data

PNO	MODE	#1	#2	#3	#4	#5	#6	#7	#8	#9	#10	#11	#12	。
*	M	a.	b.	c.	d.	e.	f.	g.	h.	i.	j.	k.	l.	

a. through l. "M CODE (MENU)?"

Select M codes from the menu. Maximum 12 M codes can be set. Setting with ten keys is also possible (up to maximum three digits).

Notice, however, that M codes which can be set differ with models.

M code menu

—00—	—01—	—03—	—04—	—05—	—08—	—09—		
PROGRAM STOP	OPTIONAL STOP	SPINDLE FORWARD	SPINDLE REVERSE	SPINDLE STOP	COOLANT ON	COOLANT OFF	CANCEL	→→→
—10—	—11—	—31—	—32—	—46—	—47—	—51—	—52—	
T/S QUIL ONLY	T/S BODY AND QUIL	TAILSTCK FORWARD	TAILSTCK BACKWARD	RANDOM OFF	RANDOM ON	ERR-DETC OFF	ERR-DETC ON	→→→
—06—	—07—	—48—	—49—	—55—	—56—	—57—	—58—	
CHUCK OPEN	CHUCK CLOSE	FWD PART CATCHER	FWD PART CATCHER	PARTS COUNT	FRNT DOR OPEN	FRNT DOR CLOSE	CHUCK AIR BLOW	→→→
—19—	—16—	—17—	—18—	—86—	—87—	—88—	—89—	
SPINDLE ORIENT	ORIENT 0°	ORIENT 120°	ORIENT 240°	WRK REST 1 OPEN	WRK REST 1 CLOSE	WRK REST 2 OPEN	WRK REST 2 CLOSE	→→→

Notes (1) M codes are executed in the order in which they were set.

Note (2) Return/not return to the tool change position after executing M process is determined by the parameter U26. However, this is to be done only when "CONTINUE" or "NUMBER" has been specified in the END process. (Refer to 3-13 Program End Data.)

3-12-2 M code table

M code	Function	Option
M00	Program stop	
M01	Optional stop	
M02	End of program (EIA/ISO program only)	*
M03	Spindle forward revolution	
M04	Spindle reverse revolution	
M05	Spindle stop	
M06	Chuck unclamp	*
M07	Chuck clamp	*
M08	Coolant on	
M09	Coolant off	
M10	Tail body joint on	
M11	Tail body joint off	
M12		
M13		
M14		
M15		
M16	Orient command 0°	*
M17	Orient command 120°	*
M18	Orient command 240°	*
M19	Orient command	*
M20	Robot service	*
M21	Robot service	*
M22	Robot service	*
M23	Robot service	*
M24	Robot service	*
M25	Robot service	*
M26	Robot service	*
M27	Robot service	*
M28	Robot service	*
M29	Robot service	*
M30	Reset, rewind (EIA/ISO program only)	*
M31	Tailstock extended	
M32	Tailstock retracted	

M code	Fuction	Option
M33	Chuck low voltage	*
M34	Chuck high voltage	*
M35		
M36	Gear shift "N"	*
M37	Gear shift "L"	*
M38	Gear shift "M"	*
M39	Gear shift "H"	*
M40	Gear shift "N"	*
M41	Gear shift "L"	*
M42	Gear shift "M"	*
M43	Gear shift "H"	*
M44	Gear shift "HH"	*
M45		
M46	Turret swing in one direction	*
M47	M46 cancel	*
M48	Parts catcher extended	*
M49	Parts catcher retracted	*
M50		
M51	Error detection off	
M52	Error detection on	
M53	Chamfering off	*
M54	Chamfering on	*
M55	Piece count	*
M56	Front door opened	*
M57	Front door closed	*
M58	Air blow	*
M59	Program completion	*
M60	Milling mode gear shift "N"	*
M61	Milling mode gear shift "L"	*
M62	Milling mode gear shift "H"	*
M63		
M64		
M65		

M code	Function	Option
M66		
M67		
M68	Cycle bar feeder call 1	*
M69	Cycle bar feeder call 2	*
M70	Spindle overload detection invalid	*
M71	Spindle overload detection valid	*
M72		
M73		
M74		
M75		
M76	Chuck jaw change spindle → removal (unload)	*
M77	Chuck jaw change spindle → carried in (load)	*
M78	Steady rest dog valid	*
M79	Steady rest dog invalid	*
M80		
M81	Work measurement start	*
M82	Work measurement end	*
M83	Tool measurement start	*
M84	Tool measurement end	*
M85		
M86	Work rest 1 unclamped	*
M87	Work rest 1 clamped	*
M88	Work rest 2 unclamped	*
M89	Work rest 2 clamped	*
M90		
M91		
M92		
M93		
M94		
M95		
M96		
M97		
M98	EIA → EIA Subprogram call (EIA/ISO program only)	*

M code	Fuction	Option
M99	EIA → EIA returned from subprogram (EIA/ISO program only)	*
M100		
M101		
M102		
M103		
M104		
M105		
M106		
M107		
M108		
M109		
M110		
M111		
M112		
M113		
M114		
M115		
M116		
M117		
M118		
M119		
M120		
M121		
M122		
M123		
M124		
M125		
M126		
M127		
M128		
M129		
M130		

M code	Function	Option
M131		
M132		
M133		
M134		
M135		
M136		
M137		
M138		
M139		
M140		
M141		
M142		
M143		
M144		
M145		
M146		
M147		
M148		
M149		
M150		
M151		
M152		
M153		
M154		
M155		
M156		
M157		
M158		
M159		
M160		
M161		
M162		
M163		

M code	Function	Option
M164		
M165		
M166		
M167		
M168		
M169		
M170		
M171		
M172		
M173		
M174		
M175		
M176		
M177		
M178		
M179		
M180		
M181	Chuck jaw No.1 (AJC)	*
M182	Chuck jaw No.2 (AJC)	*
M183	Chuck jaw No.3 (AJC)	*
M184	Chuck jaw No.4 (AJC)	*
M185	Chuck jaw No.5 (AJC)	*
M186	Chuck jaw No.6 (AJC)	*
M187	Chuck jaw No.7 (AJC)	*
M188	Chuck jaw No.8 (AJC)	*
M189	Chuck jaw No.9 (AJC)	*
M190	Chuck jaw No.10 (AJC)	*
M191	Chuck jaw No.11 (AJC)	*
M192	Chuck jaw No.12 (AJC)	*
M193	Chuck jaw No.13 (AJC)	*
M194	Chuck jaw No.14 (AJC)	*
M195	Chuck jaw No.15 (AJC)	*
M196		

M code	Function	Option
M197		
M198	Execution after EIA → MAZATROL call (EIA/ISO program only)	*
M199	Stop after EIA → MAZATROL call (EIA/ISO program only)	*

3-13 Program End Data

Select the END key from the menu.

END	MODE	COUNTER	RETURN	WK.NO.	CONTINUE	NUMBER	SFT	AMT
*	END	a.	b.	c.	d.	e.	f.	

a. "PARTS COUNT YES(1), NO(0)?"

Set "1" if the number of finished workpieces should be counted. Set "0" if it does not need to be counted.

Enter the number of workpieces to be counted as the part count limit (as a figure within parentheses in "COUNTER"). When the parts count limit has been reached, machining of workpieces is stopped. If the parts count limit is 0, this function will not work.

b. "RET POSITION(2), ZERO(1) NO(0)?"

When machining has completed:

To return the tool to the tool swing position, set "0".

To return the tool to the machine zero point, set "1".

To return the tool on the fixed point, set "2".

The tool position to which the tool is to be moved upon execution of the END mode can also be set by the user parameter P1.

Parameter P1 = 0: Return to machine zero point and fixed point is valid.

Parameter P1 = 1: Return to machine zero point and fixed point is invalid.

Position to which tool is moved after completion of machining:

Parameter P1	0	0	0	0	0	0	1	1	1	1	1	1
END mode "RETURN"	0	0	1	1	2	2	0	0	1	1	2	2
END mode "NUMBER"	0	N*	0	N*	0	N*	0	N*	0	N*	0	N*
Position to which tool is to be moved	A	A	B	D	C	E	A	A	B	B	C	C

* The position of the END mode e. is supposed to indicate the number N.

A : Return to the tool swing position set by user parameter P2

B : Return to machine zero point

C : Return to fixed position

D : Return to tool swing position set by user parameter P2 if the number is 1st to (N-1)-th. In the N-th, the tool is returned to the machine zero point.

E : The tool is returned to the tool swing position set by the user parameter P2 if the number is 1st through (N-1)-th. The tool is returned to the fixed point when the number reaches N-th.

For the tool swing position set by the user parameter P2, refer to the description about the Manual Program Mode (MNP).

• Fixed point is determined by the parameters U4 and U5.

c. "NEXT WORKPIECE NO.?"

To execute or call another work No. after machining, specify such work No. If nothing is specified, the top of the same program is called after machining.

d. "CONTINUE YES(1), NO(0)?"

If other work Nos. defined in c. are to be continuously executed after completion of the current work No., set "1". If "1" has been set, starting is performed automatically and continuous operation is executed. If "0" has been set, machining is completed after a session of machining.

e. "REPEAT TIMES OF SAME PROGRAM?"

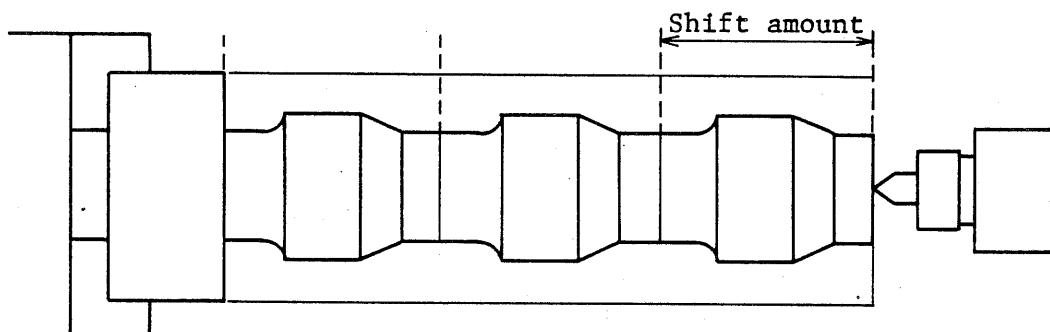
Set the number of operations in case the current work No. is to be executed repeatedly.

To complete machining when the set number of operations have been completed, set "0" in d. "CONTINUE". If "1" has been set in "CONTINUE", machining is continuously repeated, irrespective of the setting in "NUMBER".

f. "Z-AXIS SHIFT AMOUNT?"

Set the amount equivalent to the pitch in case machining of the same shape is performed by the same program or multi-piece machining is performed by special functions such as bar feeder cycle. The setting of "SFT AMT" is invalid if nothing has been set in "CONTINUE" or "NUMBER".

Machining of the same shape by the same program:



Note : Perform MEASURE process with shift amount being "0".

Especially when carrying out Z offset measurement with shift amount having been set to a certain value, the workpiece is to move deviated by that shift amount in subsequent move after it has been touched by sensor. This may result in damage to the sensor tool.

Relationships among c. WK.NO., d. CONTINUE and e. NUMBER (f. SFT AMT).

Because f. SFT AMT is invalid unless e. NUMBER is set, they must be considered as a unit.

If the end of the work No. A has been set as follows:

WK.NO.	CONTINUE	NUMBER (SFT AMT)	OPERATION
0	0	0	(1)
B	0	0	(2)
0	1	0	(3)
B	1	0	(4)
0	0	N (a)	(5)
B	0	N (a)	(6)
0	1	N (a)	(7)
B	1	N (a)	(8)

Then, the following operations will be performed:

- (1) Machining is completed after Work No. A has been executed once.
- (2) The head of the Work No. B is called after the Work No. A has been executed once.
- (3) Work No. A is executed repeatedly.
- (4) Work No. B is executed after Work No. A has been executed once.
- (5) Work No. A is executed N times with shifting equivalent to a.
- (6) The head of the program for Work No. B is called after Work No. A has been executed N times with shifting equivalent to a.
- (7) Work No. A is executed N times with shifting equivalent to a. Then, shift amount is cleared and work No. A is executed repeatedly with shifting equivalent to a.
- (8) Work No. B is executed after Work No. A has been executed N times with shifting equivalent to a.

Note : 1) In END mode, M01 (optional stop) function is not incorporated.

- 2) When "NUMBER" and "SFT AMT" are set in END mode, SHAPE CHECK and CHECK (tool path) are graphed only in the first execution.

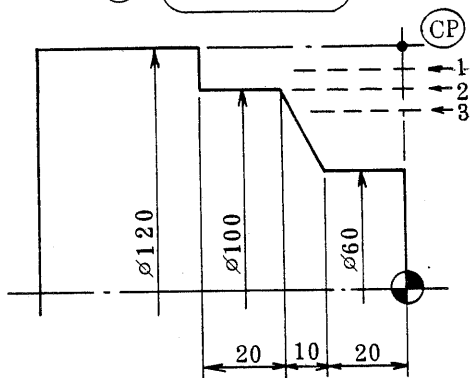
4. SIMPLE PROGRAM EXAMPLE AND TOOL PATH

Program Examples for each Mode :

In the program examples RV, FV, FEED, DEPTH and GR are omitted.

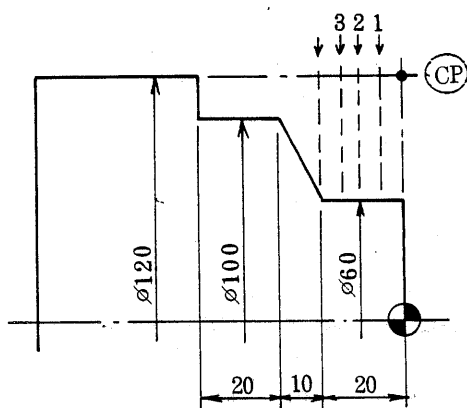
Examples 1 (BAR: OUT FCE IN)

① BAR OUT



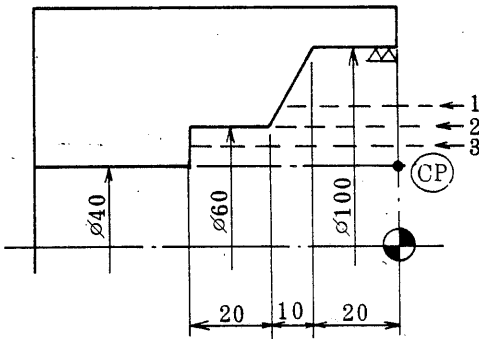
PNO	MODE	PART	CPT-X	CPT-Z				RT	FT
1	BAR	OUT	120	0				1	3
SEQ	SHP	S-CNR	SPT-X	SPT-Z	FPT-X	FPT-Z		SRF	
1	LIN				60	20		4	
2	TPR		60	20	100	30		4	
3	LIN				100	50		4	

② BAR FCE



PNO	MODE	PART	CPT-X	CPT-Z				RT	FT
1	BAR	FCE	120	0				1	3
SEQ	SHP	S-CNR	SPT-X	SPT-Z	FPT-X	FPT-Z		SRF	
1	LIN				100	50		4	
2	TPR		100	30	60	20		4	
3	LIN				60	0		4	

③ BAR IN

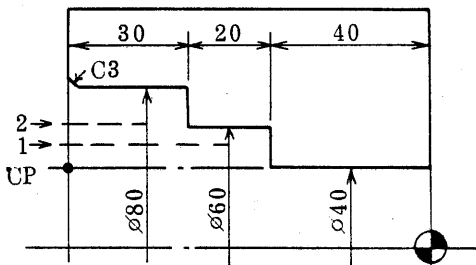


PNO	MODE	PART	CPT-X	CPT-Z			RT	FT
1	BAR	IN	40	0			1	
SEQ	SHP	S-CNR	SPT-X	SPT-Z	FPT-X	FPT-Z		SRF
1	LIN				100	20		
2	TPR		100	20	60	30		
3	LIN				60	50		

PNO	MODE	PART	CPT-X	CPT-Z			RT	FT
2	BAR	IN	40	0				3
SEQ	SHP	S-CNR	SPT-X	SPT-Z	FPT-X	FPT-Z		SRF
1	LIN				100	20		4

④ BAR IN

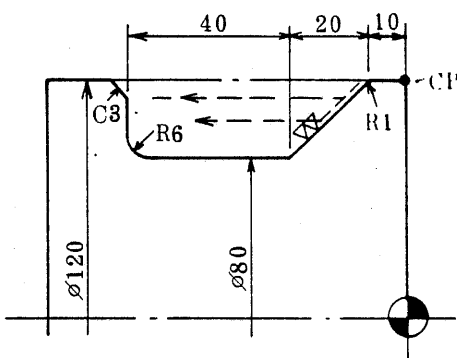
Reverse



PNO	MODE	PART	CPT-X	CPT-Z			RT	FT
1	BAR	IN	40	90			1	3
SEQ	PTN	S-CNR	SPT-X	SPT-Z	FPT-X	FPT-Z	F-CNR	SRF
1	LIN	C3			80	60		
2	LIN				60	40		

Examples 2 (BAR: FCE Reverse)

⑤ BAR

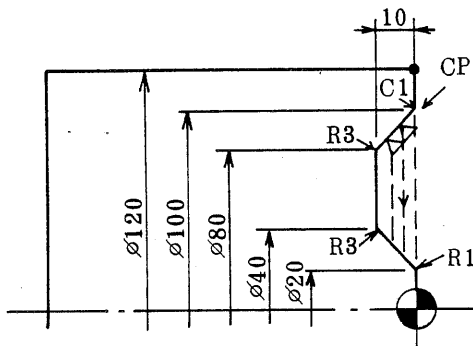


PNO	MODE	PART	CPT-X	CPT-Z			RT	FT
1	BAR	<input type="text" value="OUT"/>	120	0			1	
SEQ	SHP	S-CNR	SPT-X	SPT-Z	FPT-X	FPT-Z	F-CNR	SRF
1	TPR		120	10	80	30		
2	LIN				80	70	R6	
3	LIN	C3			120	74		

PNO	MODE	PART	CPT-X	CPT-Z			RT	FT
2	BAR	<input type="text" value="OUT"/>	120	0				3
SEQ	SHP	S-CNR	SPT-X	SPT-Z	FPT-X	FPT-Z	F-CNR	SRF
1	TPR		120	10	80	30		4

⑥

BAR FCE



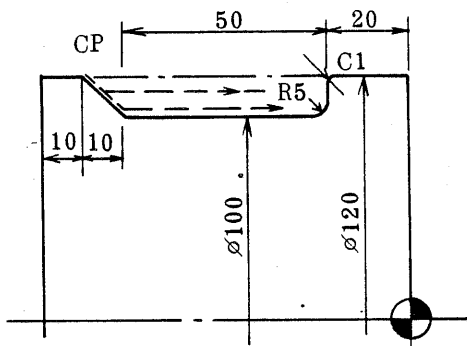
PNO	MODE	PART	CPT-X	CPT-Z			RT	FT
1	BAR	FCE	120	0			1	
SEQ	SHP	S-CNR	SPT-X	SPT-Z	FPT-X	FPT-Z	F-CNR	SRF
1	TPR	C1	100	0	80	10	R3	
2	TPR	R3	40	10	20	0	R1	
3	LIN				15	0		

PNO	MODE	PART	CPT-X	CPT-Z			RT	FT
2	BAR	FCE	120	0				3
SEQ	SHP	S-CNR	SPT-X	SPT-Z	FPT-X	FPT-Z	F-CNR	SRF
1	TPR	C1	100	0	80	10	R3	4

⑦

BAR OUT

Reverse

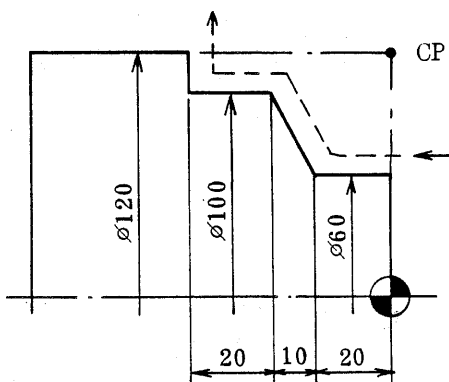


PNO	MODE	PART	CPT-X	CPT-Z			RT	FT
1	BAR	OUT	120	80			1	3
SEQ	SHP	S-CNR	SPT-X	SPT-Z	FPT-X	FPT-Z	F-CNR	SRF
1	LIN	C10			100	20	R5	
2	LIN	C1			120	18		

Examples 3 (CPY: OUT, FCE, IN)

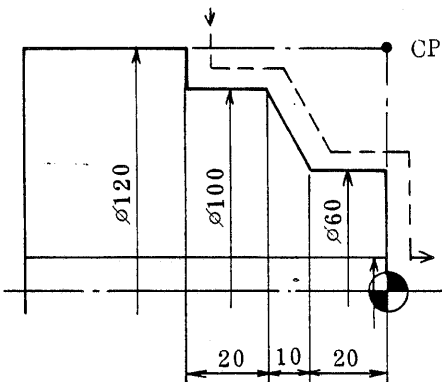
⑧

CPY OUT



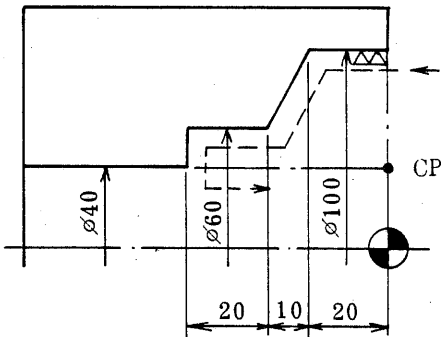
PNO	MODE	PART	CPT-X	CPT-Z	SRV-X	SRV-Z	RT	FT
1	CPY	OUT	120	0	6	3	1	3
SEQ	SHP	S-CNR	SPT-X	SPT-Z	FPT-X	FPT-Z		SRF
1	LIN				60	20		4
2	TPR		60	20	100	30		4
3	LIN				100	50		4

⑨ CPY FCE



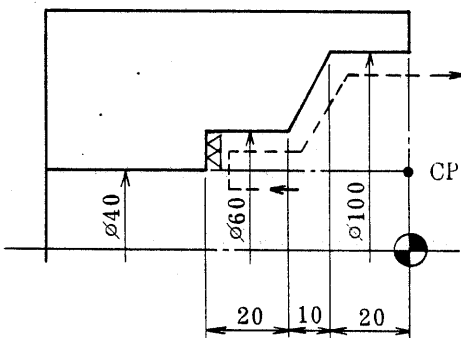
PNO	MODE	PART	CPT-X	CPT-Z	SRV-X	SRV-Z	RT	FT
1	CPY	FCE	120	0	6	3	1	3
SEQ	SHP	S-CNR	SPT-X	SPT-Z	FPT-X	FPT-Z		SRF
1	LIN				100	50		4
2	TPR		100	30	60	20		4
3	LIN				10	0		4

⑩ CPY IN



PNO	MODE	PART	CPT-X	CPT-Z	SRV-X	SRV-Z	RT	FT
1	CPY	IN	40	0	6	3	1	
SEQ	SHP	S-CNR	SPT-X	SPT-Z	FPT-X	FPT-Z		SRF
1	LIN				100	20		
2	TPR		100	20	60	30		
3	LIN				60	50		
PNO	MODE	PART	CPT-X	CPT-Z	SRV-X	SRV-Z	RT	FT
2	CPY	IN	40	0	6	3		3
SEQ	SHP	S-CNR	SPT-X	SPT-Z	FPT-X	FP-Z		SRF
1	LIN				100	20		4

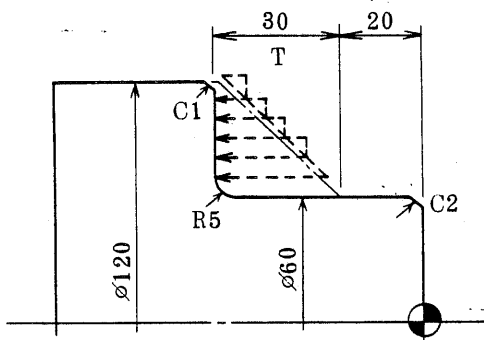
⑪ CPY FCE



PNO	MODE	PART	CPT-X	CPT-Z	SRV-X	SRV-Z	RT	FT
1	CPY	FCE	40	0	6	3	1	
SEQ	SHP	S-CNR	SPT-X	SPT-Z	FPT-X	FPT-Z		SRF
1	LIN				60	50		
2	TPR		60	30	100	20		
3	LIN				100	0		
PNO	MODE	PART	CPT-X	CPT-Z	SRV-X	SRV-Z	RT	FT
2	CPY	FCE	40	0	6	3		3
SEQ	SHP	S-CNR	SPT-X	SPT-Z	FPT-X	FPT-Z		SRF
1	LIN				60	50		4

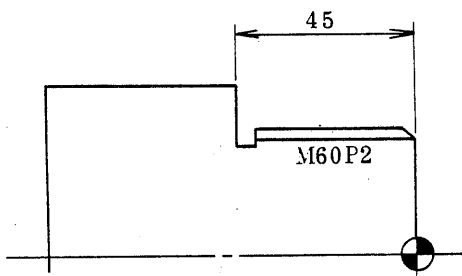
Examples 4 (CNR, THR, GRV, CPY)

⑫ CNR OUT



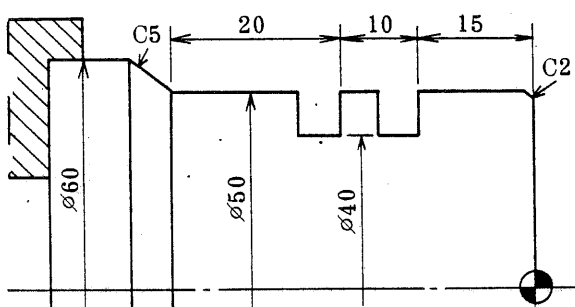
PNO	MODE	PART						RT	FT
1	CNR	OUT						1	
SEQ			SPT-X	SPT-Z	FPT-X	FPT-Z	F-CNR	SRF	
1			60	20	120	50	R5		
PNO	MODE	PART	CPT-X	CPT-Z				RT	FT
2	BAR	OUT	120	0					3
SEQ	SHP	S-CNR	SPT-X	SPT-Z	FPT-X	FPT-Z	F-CNR	SRF	
2	LIN	C2			60	50	R5		4
3	LIN	C1			120	52			4

⑬ THR OUT



PNO	MODE	PART	CHAMF	LEAD		ANG	MULTI	T
1	THR	OUT	1	2		0	1	3
SEQ			SPT-X	SPT-Z	FPT-X	FPT-Z		
1			60	0	60	43		

⑭ GRV OUT

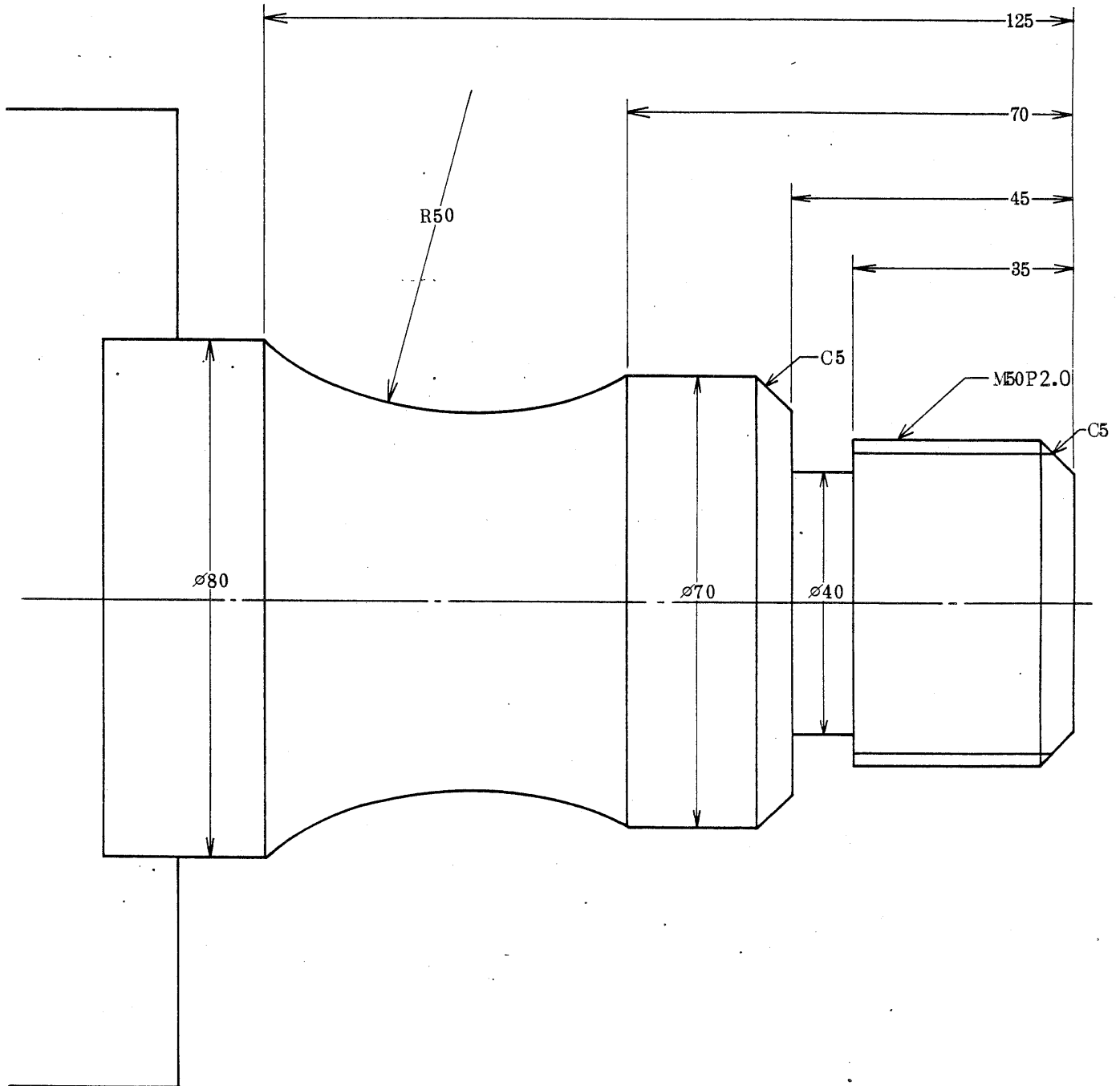


PNO	MODE	PART	#	NO.	PITCH	GRV WID	FINISH	T
1	GRV	OUT	0	2	10	5	0	4
SEQ			SPT-X	SPT-Z	FPT-X	FPT-Z		
1			50	20	40	20		

The width of the grooving tool is 5 mm.

5. PROGRAM EXAMPLE IN CUTTING WORK

① Example 1



Finish ∇ Material: CBN STL Material dimensions: $\phi 80 \times 150$

PNO	MAT	OD-MAX	ID-MIN	LENGTH	RPM	FIN-X	FIN-Z	WORK	FACE
0	CB ST	80	0	150	2000	0.4	0.1	1	

PNO	MODE	RT	FT
1	EDG	1-0	2-0

SEQ	SPT-X	SPT-Z	FPT-X	FPT-Z	SRF	\$
1	80	1	0	0	3	

PNO	MODE	PART	CPT-X	CPT-Z	RT	FT
2	BAR	OUT	80	0	3-0	4-0

SEQ	SHP	S-CNR	SPT-X	SPT-Z	FPT-X	FPT-Z	F-CNR	RADIUS	SRF	\$
1	LIN	C5			50	45	C0		3	
2	LIN	C5			70	70	C0		3	
3	<input type="checkbox"/> L	C0	70	70	80	125	C0	50	3	

PNO	MODE	PART	#	NO.	PITCH	GRV WID	FINISH	RT	FT
3	GRV	<input type="checkbox"/> OUT	0	1	0	10	0		6-0

SEQ	SHP	S-CNR	SPT-X	SPT-Z	FPT-X	FPT-Z	F-CNR	RADIUS	SRF	\$
1		C0	50	45	40	45	C0		3	

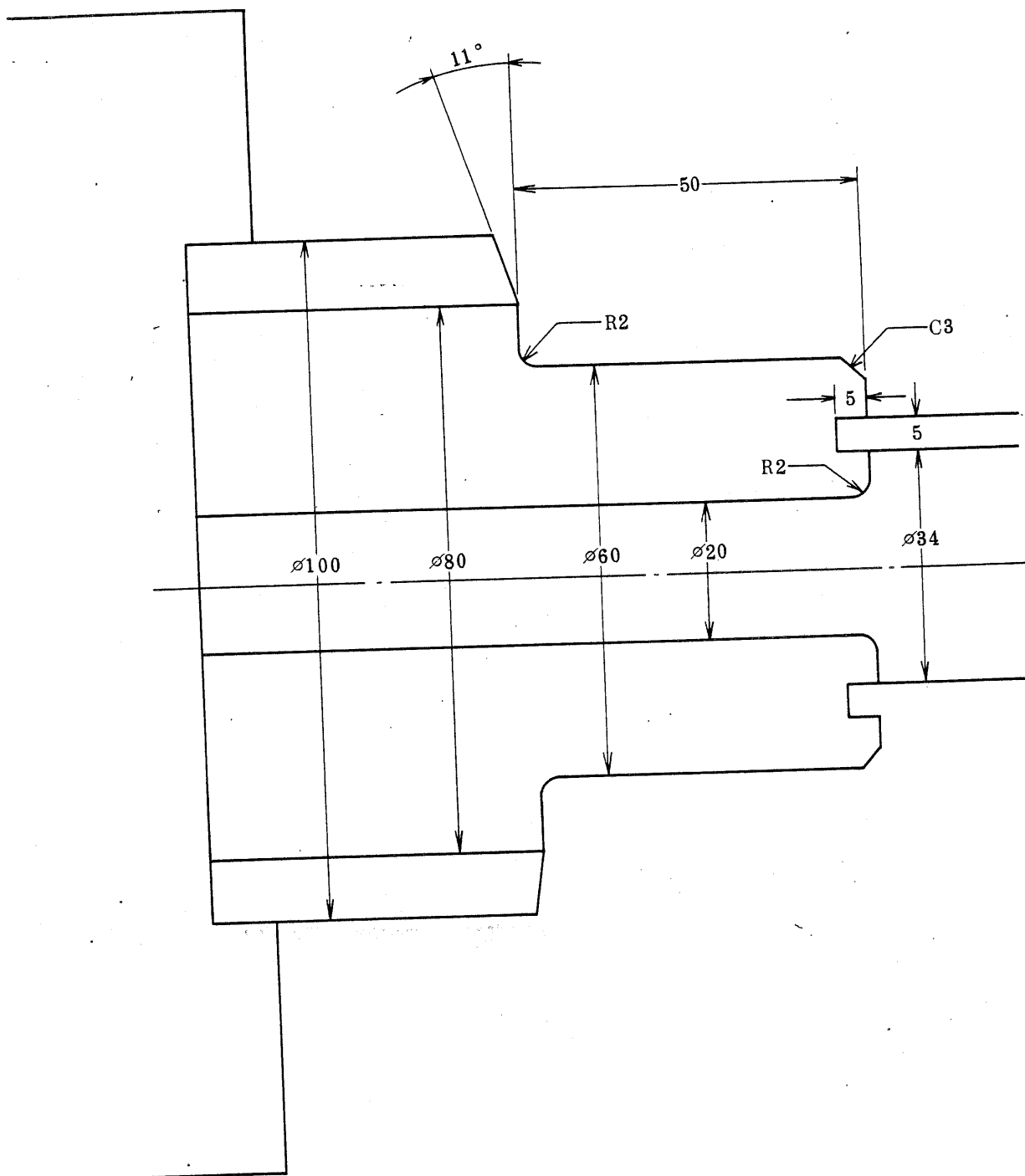
PNO	MODE	PART	CHAMF	LEAD	ANG	MULTI	HGT	NUMBER	T
4	THR	OUT	0	2	0	1	1.299	10	7-0

SEQ	SPT-X	SPT-Z	FPT-X	FPT-Z
1	50	0	50	38

PNO	MODE	COUNTER	RETURN	WK.NO.	CONTINUE	NUMBER
5	END	0	0	0	0	0

*** WORK PROGRAM NO. 1 ***

② Example 2



Finish: ∇

Material: CBN STL

Material dimensions: $\phi \times 150 \text{ L}$

PNO	MAT	OD-MAX	ID-MIN	LENGTH	RPM	FIN-X	FIN-Z	WORK	FACE
0	CB ST	100	0	150	2000	0.4	0.1	1	

PNO	MODE	RT	FT
1	EDG	1-1	2-2

SEQ	SPT-X	SPT-Z	FPT-X	FPT-Z	SRF	\$
1	100	1	0	0	3	

PNO	MODE	PART	CPT-X	CPT-Z	RT	FT
2	BAR	OUT	100	0	3-1	4-1

SEQ	SHP	S-CNR	SPT-X	SPT-Z	FPT-X	FPT-Z	F-CNR	RADIUS	SRF	\$
1	LIN	C3			60	50	R2		3	
2	TPR	C0	80	50	100	?	C0	79	3	

PNO	MODE	#	DRL DIA	DEP-1	DEP-2	DEP-3	T
3	DRL	1	18	18	5	10	5-0

SEQ	SPT-X	FPT-Z
1	0	156

PNO	MODE	PART	CPT-X	CPT-Z	RT	FT
4	BAR	IN	18	0	6-0	7-0

SEQ	SHP	S-CNR	SPT-X	SPT-Z	FPT-X	FPT-Z	F-CNR	RADIUS	SRF	\$
1	LIN	R2			20	100	C0		3	

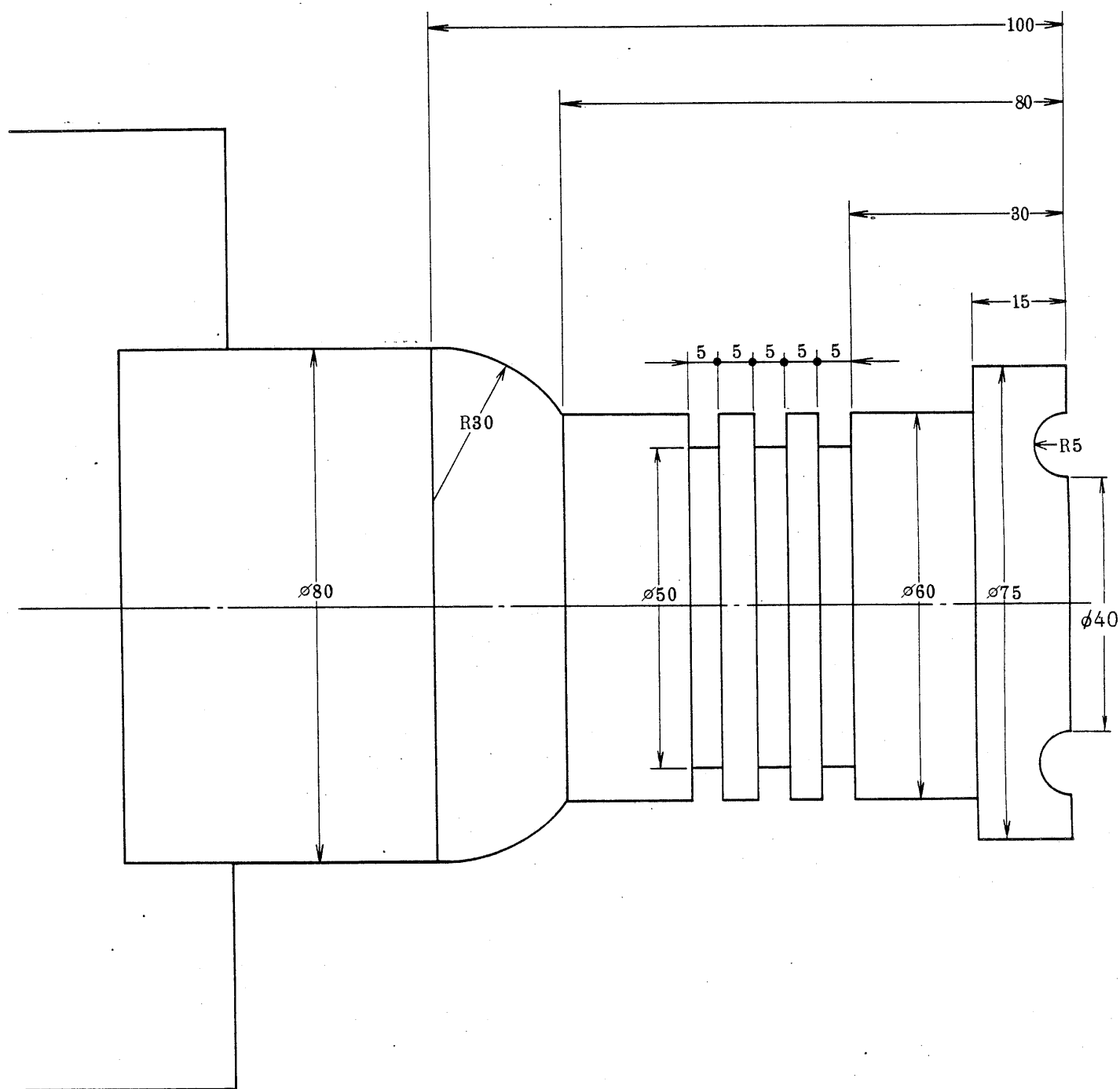
PNO	MODE	PART	#	NO.	PITCH	GRV WID	FINISH	RT	FT
5	GRV	FCE	0	1	0	5	0		8-1

SEQ	SHP	S-CNR	SPT-X	SPT-Z	FPT-X	FPT-Z	F-CNR	RADIUS	SRF	\$
1		C0	34	0	34	5	C0			

PNO	MODE	COUNTER	RETURN	WK.NO.	CONTINUE	NUMBER
6	END	0	0	0	0	0

*** WORK PROGRAM NO. 2 ***

③ Example 3




Finish: ∇ Material: CBN STL Material dimensions: $\phi \times 150 \ell$


PNO	MAT	OD-MAX	ID-MIN	LENGTH	RPM	FIN-X	FIN-Z	WORK FACE
0	CB ST	80	0	150	2000	0.4	0.1	1

PNO	MODE	RT	FT
1	EDG	1-1	2-2


SEQ	SPT-X	SPT-Z	FPT-X	FPT-Z	SRF	\$
1	80	1	0	0	3	

PNO	MODE	PART	CPT-X	CPT-Z	RT	FT
2	CPY	OUT	80	0	3-2	4-1


SEQ	SHP	S-CNR	SPT-X	SPT-Z	FPT-X	FPT-Z	F-CNR	RADIUS	SRF	\$
1	LIN	CO			75	15	CO		3	
2	LIN	CO			60	80	CO		3	
3		CO	60	80	80	100	CO	30	3	


PNO	MODE	PART	RT	FT
3	CNR		5-1	6-2

SEQ	SPT-X	SPT-Z	FPT-X	FPT-Z	F-CNR	SRF	\$
1	60	30	75	15	CO	3	

PNO	MODE	PART	#	NO.	PITCH	GRV WID	FINISH	RT	FT
4	GRV		1	3	10	5	0.1	7-1	8-2

SEQ	SHP	S-CNR	SPT-X	SPT-Z	FPT-X	FPT-Z	F-CNR	RADIUS	SRF	\$
1		CO	60	30	50	30	CO		3	

PNO	MODE	PART	CPT-X	CPT-Z	RT	FT
5	BAR		75	0	9-0	10-0

SEQ	SHP	S-CNR	SPT-X	SPT-Z	FPT-X	FPT-Z	F-CNR	RADIUS	SRF	\$
1		CO	60	0	40	0	CO	5	3	

PNO	MODE	COUNTER	RETURN	WK.NO.	CONTINUE	NUMBER
6	END	0	0	0	0	0

*** WORK PROGRAM NO.3 ***

6. FUNCTIONS

6-1 Automatic Machining Condition Determining Function and Material Registration Function

The preparation of the MAZATROL Program allows automatic determination of various machining conditions. Elements and modes subject to automatic determination are shown in the tables below:

Cutting (T-2 specification)

Cutting condition Machining mode	RV (m/min)	FV (m/min)	FEED (mm/rev)	DEPTH (mm)	HGT (mm)	NUMBER
BAR	o	o	o	o		
CPY	o	o	o	o		
CNR	o	o	o	o		
EDG	o	o	o	o		
THR	o			o	o	o
GRV	o	o	o	o		
DRL	o		o	o		
TAP	o					
SGL						

Elements for the Automatic Determination of Machining Conditions

There are two methods of automatically determining machining conditions:

- 1) Determination based on machining conditions of CBN STL material registered on the CUTTING CONDITION picture
- 2) Methods based on calculations from data and parameters set by the program

Data thus determined can be changed freely by manual operation.

6-1-1 Elements determined by CUTTING CONDITION picture

The following machining conditions are automatically determined based on the data set on the CUTTING CONDITION picture NO. 1 and NO. 2:

Surface speed or roughing surface speed (m/min.)

$RV = (\text{CUTTING CONDITION picture RV}) * (\% \text{ determined by MATERIAL on CUTTING CONDITION picture})$

Applicable machining modes:

BAR, CPY, CNR, EDG, THR, GRV, DRL, TAP,

Finish surface speed (m/min.)

$FV = (\text{CUTTING CONDITION picture FV}) * (\% \text{ determined by MATERIAL on CUTTING CONDITION picture})$

Applicable machining modes:

BAR, CPY, CNR, EDG, GRV

Feedrate in rough cutting (mm/rev.)

$FEED = (\text{FEED on CUTTING CONDITION picture}) * (\% \text{ determined by MATERIAL on CUTTING CONDITION picture})$

Applicable machining modes:

BAR, CPY, CNR, EDG, GRV, DRL

DEPTH in Rough Cutting (mm)

$DEPTH = (\text{DEPTH on CUTTING CONDITION picture}) * (\% \text{ determined by MATERIAL on CUTTING CONDITION picture})$

Applicable machining modes:

BAR, CPY, CNR, EDG, GRV

Notes:

- (1) DEPTH in finish machining is determined by the finishing allowance stored in the program.
- (2) If SRF is selected from the menu on the WORK PROGRAM picture, the following feedrate is provided:

(Ex.) Nose R: 0.8mm

SRF code	\$ ≠ 4	\$ = 4
-	Feedrate in rough machining	2000 mm/min
1	0.8 (mm/rev)	1000
2	0.565	720
3	0.4	520
4	0.282	370
5	0.2	270
6	0.143	200
7	0.101	140
8	0.071	100
9	0.05	72

Applicable machining modes: BAR, CPY, CNR, EDG, GRV

F: Feedrate

General formula $F = \sqrt{\frac{8R\mu}{1000}}$ mm/rev

R: Nose R (mm)

μ : Surface roughness (μm)

6-1-2 Elements determined by program data and parameters

The machining conditions in the following modes are automatically determined by the set data such as the program data and parameters set on the CUTTING CONDITION picture.

(1) Drilling : DRL

(i) DEP-1

The amount of first cut in drilling is determined by the following equation:

$$(DEP-1) = (\text{Drill diameter}) * (\text{parameter } \boxed{U51}) / 100$$

$\boxed{U51}$: Drilling amount coefficient

Note : DEPTH data on the CUTTING CONDITION picture does not become effective.

(ii) DEP-2

The data set by the parameter $\boxed{U49}$ is set as the decrement value of cutting amount.

$\boxed{U49}$: Drilling amount decrement

(iii) DEP-3

The data set by the parameter $\boxed{U53}$ is set as the minimum cutting amount

$\boxed{U53}$: Minimum drilling amount

(iv) FEED

(a) When (drill diameter set by program) < (parameter $\boxed{U65}$) :

$$\text{FEED} = (\text{FEED on CUTTING CONDITION picture}) * \text{PROGRAM picture} / (\text{Parameter } \boxed{U65})$$

(b) When (drill diameter set by program) \geq (parameter $\boxed{U65}$) :


$$\text{FEED} = (\text{FEED on CUTTING CONDITION picture})$$

$\boxed{U65}$: Reference diameter for automatic determination of drill feedrate

6-1-3. Method of registration of machining conditions

Data on machining conditions are registered before shipping. However, the User can perform setting in any desired manner. To change the settings, practice the following:

A. Change of Data on the ***CUTTING CONDITION NO. 1*** picture

- (1) Push the  (display page selection) key and then push **C-COND**.
- (2) Then, the CUTTING CONDITION NO. 1 picture will appear. The data set on this picture are for the CBN STL material. Move the cursor to the position where data is to be changed. Then, set the data with ten keys.

MODE	PART	SURFACE SPEED		FEED	DEPTH
		RV	FV		
BAR	OUT	130	180	0.35	3
	IN	110	150	0.3	3
	FCE	130	180	0.3	3
CPY	OUT	130	180	0.32	3
	IN	110	160	0.32	3

Push **1** **6** **5** **INPUT** in this order.

MODE	PART	SURFACE SPEED		FEED	DEPTH
		RV	FV		
BAR	OUT	130	180	0.35	3
	IN	110	150	0.3	3
	FCE	130	165	0.3	3
CPY	OUT	130	180	0.32	3
	IN	110	160	0.32	3


Notes:











- (1) The surface speed for THR (threading), DRL (drilling) and TAP (tapping) is only effective for RV.
- (2) Feedrate for THR and TAP is invalid.
- (3) Cutting amount for DRL (drilling) is set by the parameter U51.
- (4) Cutting amount can be set up to the third lower digit than the decimal point (in mm).


B. Change of Data on the *CUTTING CONDITION NO. 2*** picture**

- (1) Push the PAGE on the CUTTING CONDITION NO. 1 picture.
- (2) Then, the CUTTING CONDITION NO. 2 picture is displayed. Percentage for each material type (MATERIAL) is set in data on this picture.

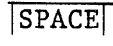

Move the cursor to the position where the registered data is to be changed. Set new data with menu keys or ten keys.

No.	MATERIAL	%
1	CB ST	100
2	AL ST	80
:	:	
:	:	
6		
7		

Push  six times and then push     
 (INPUT) key    (INPUT) key in this order.

NO.	MATERIAL	%
1	CB ST	100
2	AL ST	80
:	:	
:	:	
6	FC21M	78
7		

Notes:

- (1) On this picture, the User can set data on MATERIAL No. 6 through No. 16. Data on No. 1 through No. 5 are set by YAMAZAKI.
- (2) To erase the material type, push the  and  (INPUT) keys.
- (3) Setting by the material type set by the user on the WORK PROGRAM picture is impossible. Instead, set by a MAT No.

6-2 Program Edit Function

The information in a program may easily be altered, reproduced and retrieved by use of the following EDIT functions:


1. PROCESS NO. SEARCH A process number, if entered, will be displayed on the picture. This convenient function is used to retrieve desired data in a long program.
2. LAST SEARCH The end of a program is displayed on the picture.
3. PROCESS COPY A certain process and the related sequence data in the program prepared or being prepared is inserted into a selected portion and copied.
4. PROGRAM COPY ... A prepared program is copied into the program being prepared.
5. INSERT A program or sequence data is inserted on a line by line basis.
6. ERASE Data are erased on a process by process or sequence by sequence basis.

Note : All of the above operations are available only in MAZATROL Programs. For information on EIA/ISO programs (optional), see the EIA/ISO PROGRAMMING MANUAL.

6-2-1 PROCESS NO. SEARCH

To retrieve certain data contained in a long program, enter the process number, this will be displayed on the picture.


Operation

1 While preparing a program, press the  (menu select key).

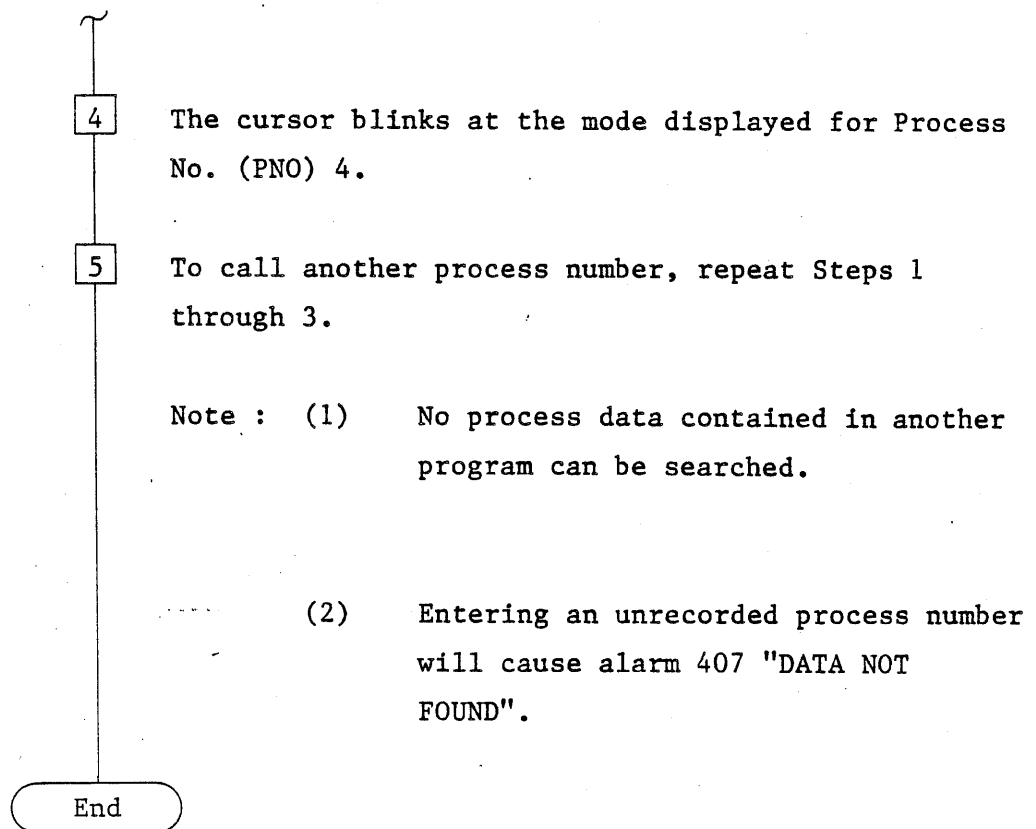
2 Press PROC NO. SEARCH.

PNO	MAT	O.D. MAX	I.D. MAX
0	CB ST	100	60.5
PNO	MODE	PART	CPT-X
1	BAR	OUT	60.5
SEQ	SHP	S-CNR	
1	LIN	C1.5	
2	LIN	C0	

3 In response to "PROCESS NUMBER?", set the process No. (PNO) to be searched using ten keys.

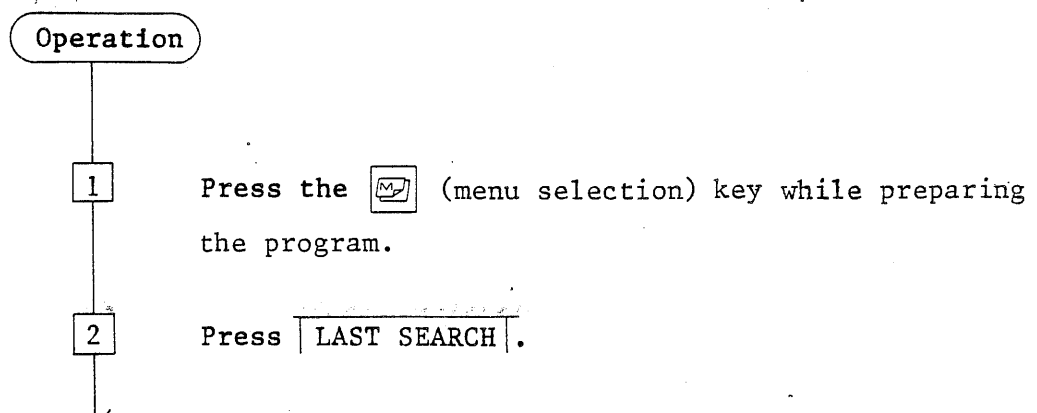
For example, press 4 and  (INPUT) key.

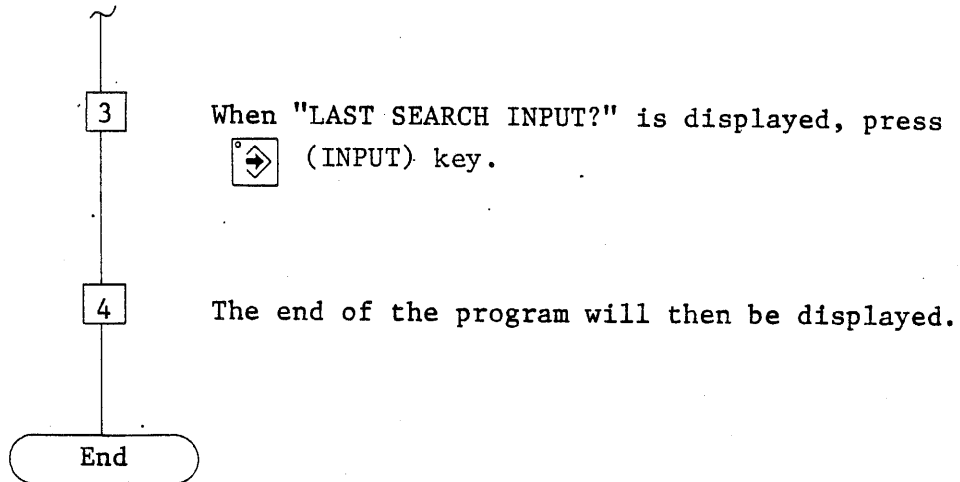
SEQ	SHP	S-CNR	SPT-X
1	LIN	C1.0	
PNO	MODE	PART	CHAMF
4	THR	OUT	1
SEQ			SPT-X
1			40



6-2-2 LAST SEARCH

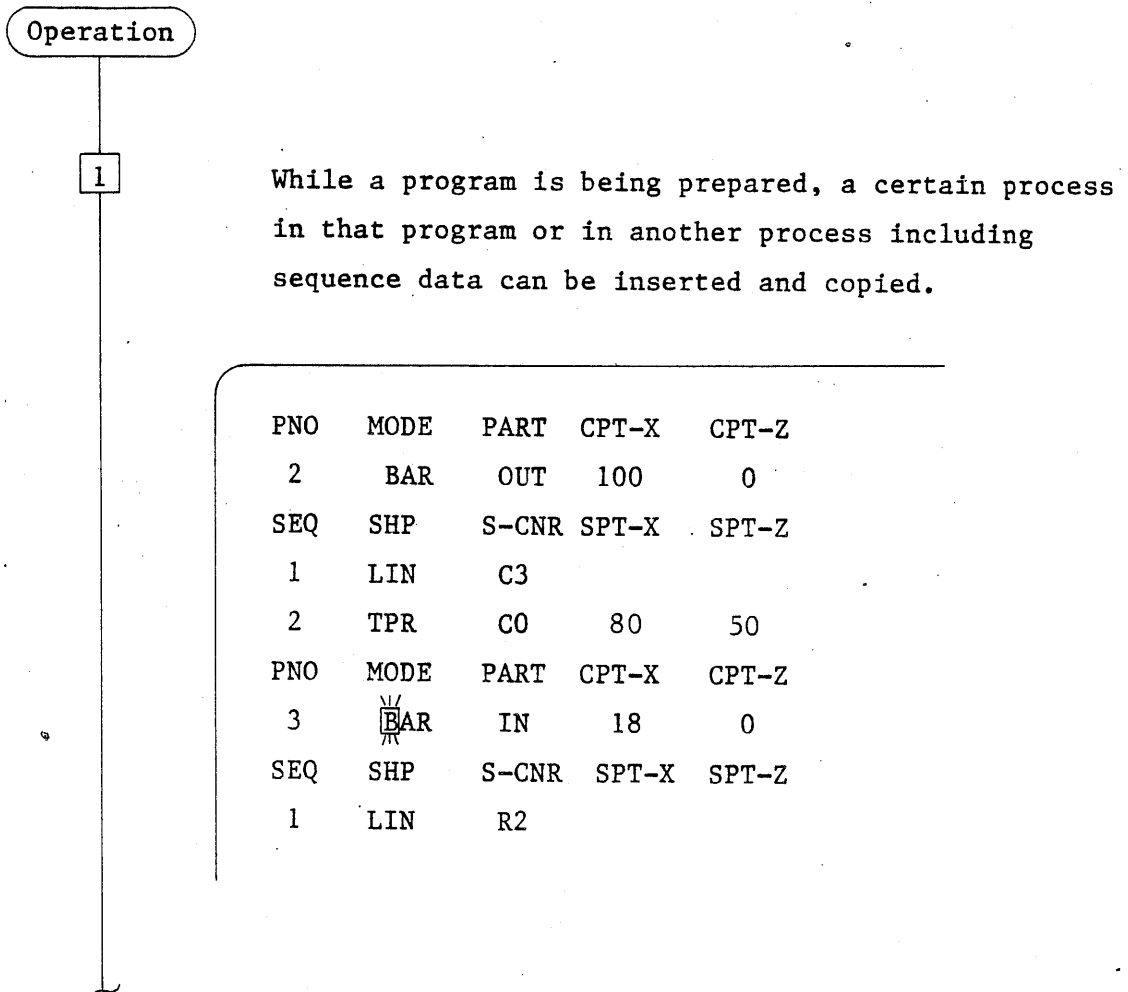
It is convenient when preparing programs one after another since the program end can be called while the program is being prepared.






Note : If program has an END mode, the LAST SEARCH function will call it automatically.

6-2-3 PROCESS COPY








2

Press  and PROCESS COPY in that order.

3



In response to "WORKPIECE NO.?", use ten keys to enter the work number whose process is to be copied.

If the work number goes 1234, press , , ,  and  (INPUT) key.

Note: (1) The work number of a program prepared or being prepared should be entered. Entering an unrecorded work number, however, will result in alarm 407 "DATA NOT FOUND".

(2) If the cursor is positioned out of the process, it will result in alarm 475 "CURSOR POSITION IN ADEQUATE".

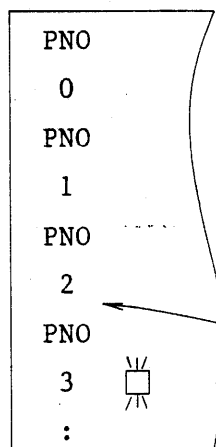
4

In response to "PROCESS NUMBER?", use ten keys to enter the process number to be copied and inserted. If the process number (PNO) goes, 4, press  and  (INPUT) key.

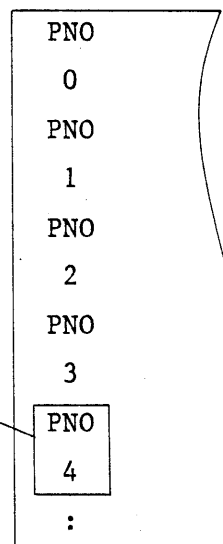
5

Work No. 1234, PNO 4, will be inserted and copied prior to PNO 3, thus serially numbering the process.

Work No. 100

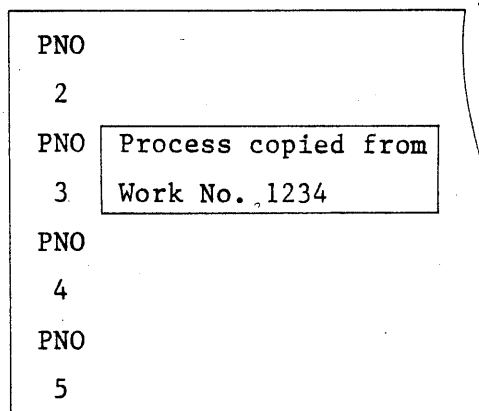


Work No. 1234



PROCESS COPY

Work No. 100



End

- Note : (1) PNO. 0 common data are uncopiable.
An attempt to copy PNO. 0 will result in alarm 412
"ILLEGAL PROCESS COPY".
- (2) A process in the END mode is copiable. However, do not perform this copying, because such cannot be executed correctly if machining data are available from the END mode on.
- (3) A process in the EIA program is uncopiable. An attempt to copy an EIA process will result in alarm 418 "EIA PROGRAM IS DESIGNATED".
- (4) A process in the MTR is uncopiable. An attempt to copy a MTR process will result in alarm 412 "ILLEGAL PROCESS COPY".
- (5) No process is copiable prior to PNO. 0 common data. An attempt to do so will result in alarm 475 "CURSOR POSITION INADEQUATE".

6-2-4 PROGRAM COPY

Another program can be inserted into the program being prepared.


Operation

1

While preparing the program, WNo. 100
move the cursor to the position where another program is to be copied. (This operation is similar to that for PROCESS COPY.)

PNO	
0	CS IR
PNO	
1	BAR IN
PNO	
2	GRV OUT

2






Press  and **PROGRAM COPY** in that order.

Program to be copied

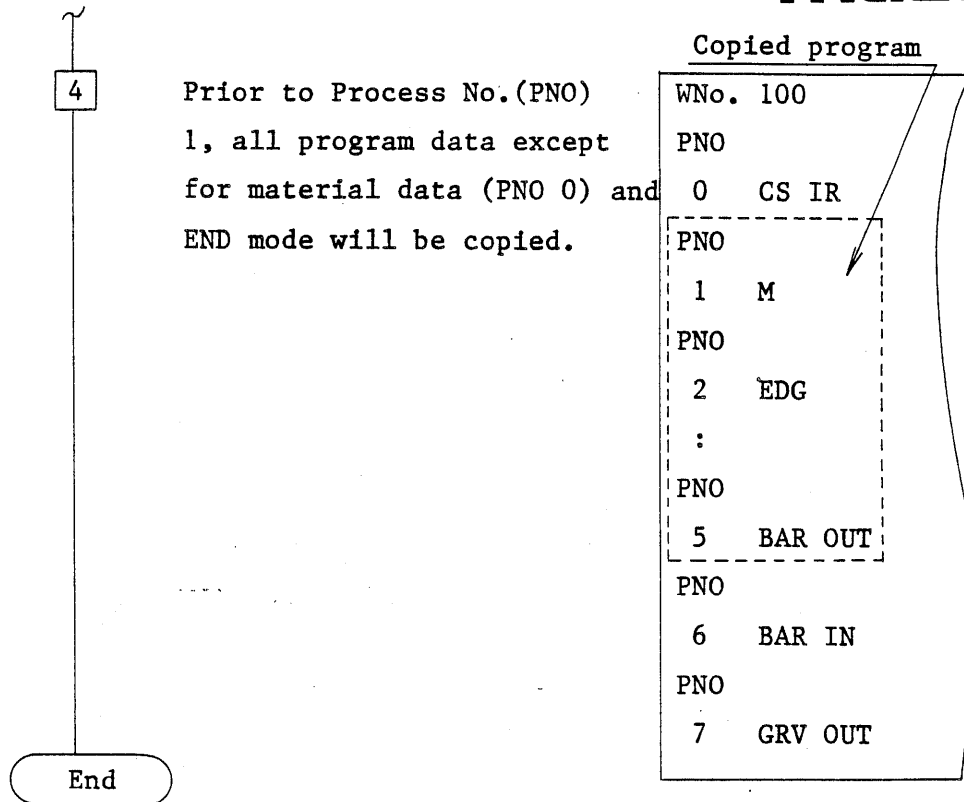
WNo. 1234

3

In response to "WORKPIECE NUMBER", use ten keys to enter the work number to be copied.

If the work number goes 1234, press , , ,  and  (INPUT) key.

PNO	
0	CB ST
PNO	
1	M
PNO	
2	EDG
:	
PNO	
5	BAR OUT
PNO	
6	END



- Note: (1) The program being currently prepared is uncopiable.
Attempting to do so will result in alarm 415 "SAME PROGRAM IS DESIGNATED".
- (2) Unless the cursor position is available in the process, it will result in alarm 475 "CURSOR POSITION INADEQUATE".
- (3) It is not possible to copy the programs registered in the bubble memory.
Attempting to do so will result in alarm 407 "DATA NOT FOUND".
(optional)
Likewise, the program not registered is also uncopiable.
- (4) Any EIA program is uncopiable. Attempting to do so will result in alarm 418 "EIA PROGRAM DESIGNATED".
Prior to PNO. 0 common data, no program is copiable.
Attempting to do so will result in alarm 475 "CURSOR POSITION INADEQUATE".
- (5) Prior to PNO.0 common data, no program is copiable.
Attempting to do so will result in afarm 475 "CURSOR POSITION IADEQUATE".


6-2-5 INSERT

While preparing a program, it is possible to insert processes or sequence data on a line by line basis.


Operation

1


While preparing the program, move the cursor to the next block to the line to the line to be inserted. Within an identical block, the cursor may be positioned anywhere except PNO. 0.


PNO	MAT	OD-MAX
0	CB ST	100
PNO	MODE	PART
1	BAR	OUT
SEQ	SHP	S-CNR
1	 LIN	C1.5
2	LIN	C0

2

Press  (menu selection) key and INSERT in that order.

3

In response to "INSERT (INPUT)?", press  (INPUT) key. Then, one line of sequence data can be inserted.

PNO	MAT	OD-MAX
0	CB ST	100
PNO	MODE	PART
1	BAR	OUT
SEQ	SHP	S-CNR
1		
2	LIN	C1.5
3	LIN	C0

4

To insert several lines of data in succession, repeat Step 2 and 3 above.

End

Note: (1) Carry out the INSERT operation next to PNO. 0 common data and it will be possible to insert the process data.

- (2) It is impossible to carry out the INSERT operation for the sequence data specified below. Attempting to do so will result in alarm 475 "CURSOR POSITION INADEQUATE":

CRN, EDG, GRV, DRL, TAP

6-2-6 ERASE

It is possible to erase process or sequence data on a process by process or sequence by sequence basis.

- A** To erase process data on a line by line basis:


Operation

1

While preparing the program, move the cursor to the sequence data desired to be erased.


PNO	MODE	PART	CPT-X
2	BAR	OUT	100
SEQ	SHP	S-CNR	SPT-X
1	<u>LIN</u>	C1.5	
2	TPR	C0	80
PNO	MODE	PART	CPT-X
3	BAR	IN	35
SEQ	SHP	S-CNR	SPT-X
1	LIN	C1.5	

2

Press  (menu selection) key and **ERASE** in that order.

PNO	MODE	PART	CPT-X
2	BAR	OUT	100
SEQ	SHP	S-CNR	SPT-X
1	TPR	C0	80
PNO	MODE	PART	CPT-X
3	BAR	IN	35
SEQ	SHP	S-CNR	SPT-X
1	LIN	C1.5	

3


In response to "ERASE (INPUT)?", press the  (INPUT) key. The sequence data pointed at by the cursor will be erased by one line.


End

B To erase all of the process data and sequence data:

Operation

1 While preparing the program, move the cursor to the process data to be erased.

2 Press  (menu selection) key and **ERASE** in that order.

3 In response to "ERASE (INPUT)?", press  (INPUT)key. All of the process selected with the cursor and sequence data thereof will be erased.

End

```
PNO MODE PART CPT-X
2  BAR OUT 100
SEQ SHP S-CNR SPT-X
1  LIN C1.5
2  TPR CO 80
PNO MODE PART CPT-X
3  BAR IN 35
SEQ SHP S-CNR SPT-X
1  LIN C1.5
```

```
PNO MODE PART CPT-X
2  BAR IN 35
SEQ SHP S-CNR SPT-X
1  LIN C1.5
PNO MODE COUNTER RETURN
3  END 1 0
```

Note: PNO. 0 common data are unerasable. Attempting to erase PNO. 0 common data will result in alarm 475 "CURSOR POSITION INADEQUATE". When PNO. 0 common data are to be erased, perform the program erase operation.

6-3 LAYOUT FUNCTION (Available in MAZATROL Programs only)


Programs prepared may be easily laid out (to edit machining processes).

The EDIT function is available in the following three types:

1. Program priority machining,
2. Rough priority machining and
3. Editing processes in order.

The operation should be carried out on the LAYOUT picture.

6-3-1 Displaying the LAYOUT Picture

Press  (picture select) key, **PROGRAM** and **LAYOUT** in that order and the LAYOUT picture will be displayed:

NO.	PNO.	PROCESS	TNO.	TIME
1	R 1	BAR-OUT	1-0	0: 1' 2"
2	F 1	BAR-OUT	2-1	0: 0'43"
3	R 2	BAR-IN	4-0	0: 1'38"
4	F 2	BAR-IN	6-2	0: 0'57"

TOTAL CUTTING TIME 0:13'21"

***LAYOUT NO. ***

ROUGH PRIORITY	PROGRAM PRIORITY	REPLACE	MOVE	ROUGH R	FINISH F	CHECK	PROGRAM	SIMULATION
-------------------	---------------------	---------	------	------------	-------------	-------	---------	------------

└─ Machining mode

- (3) TNO. representing the number of a tool to be used in machining and its offset number.

$\begin{array}{c} \text{1} \quad - \quad \text{0} \\ \text{└─} \quad \text{└─} \\ \text{Tool number} \quad \text{Offset number} \end{array}$

- (4) TIME representing the time required for machining in each process after executing the simulation.

- (5) TOTAL CUTTING TIME ... The total machining time is displayed in the lower part of the program after executing the simulation.

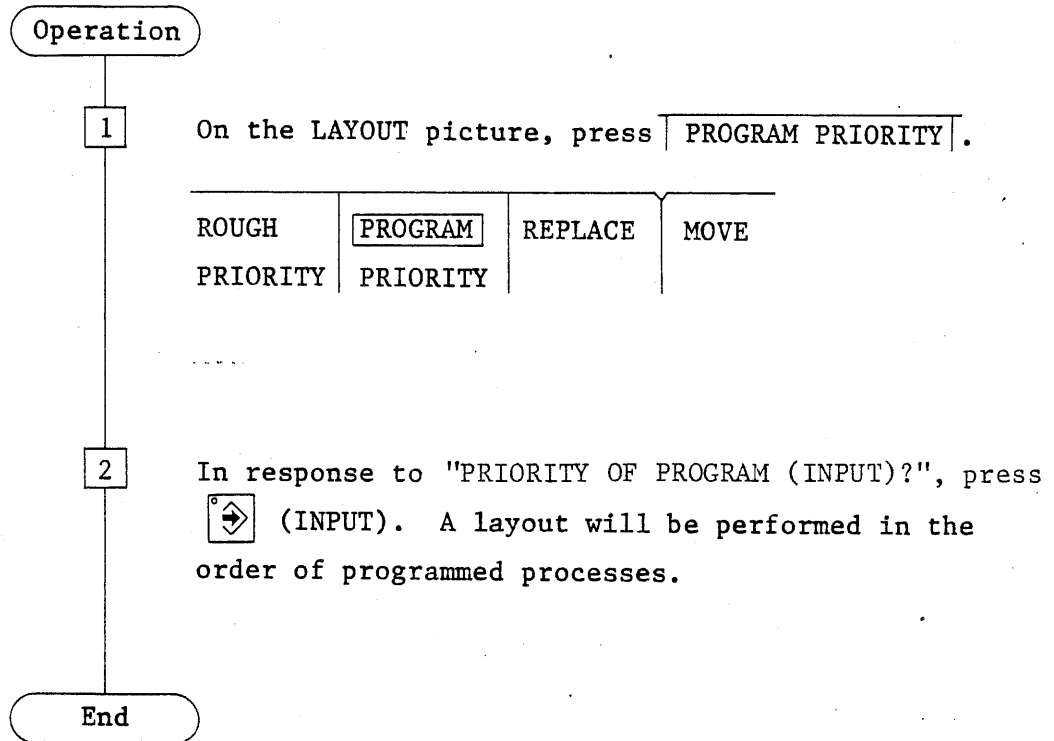
6-3-2 LAYOUT operation

1 PROGRAM PRIORITY

This PROGRAM PRIORITY layout will be convenient to carry out machining subject to the programmed process sequence after preparing a MAZATROL Program.

At the same time when the MAZATROL Program is prepared, the PROGRAM priority machining will be automatically laid

out. To lay out the program priority after editing a certain layout, carry out the following operation:

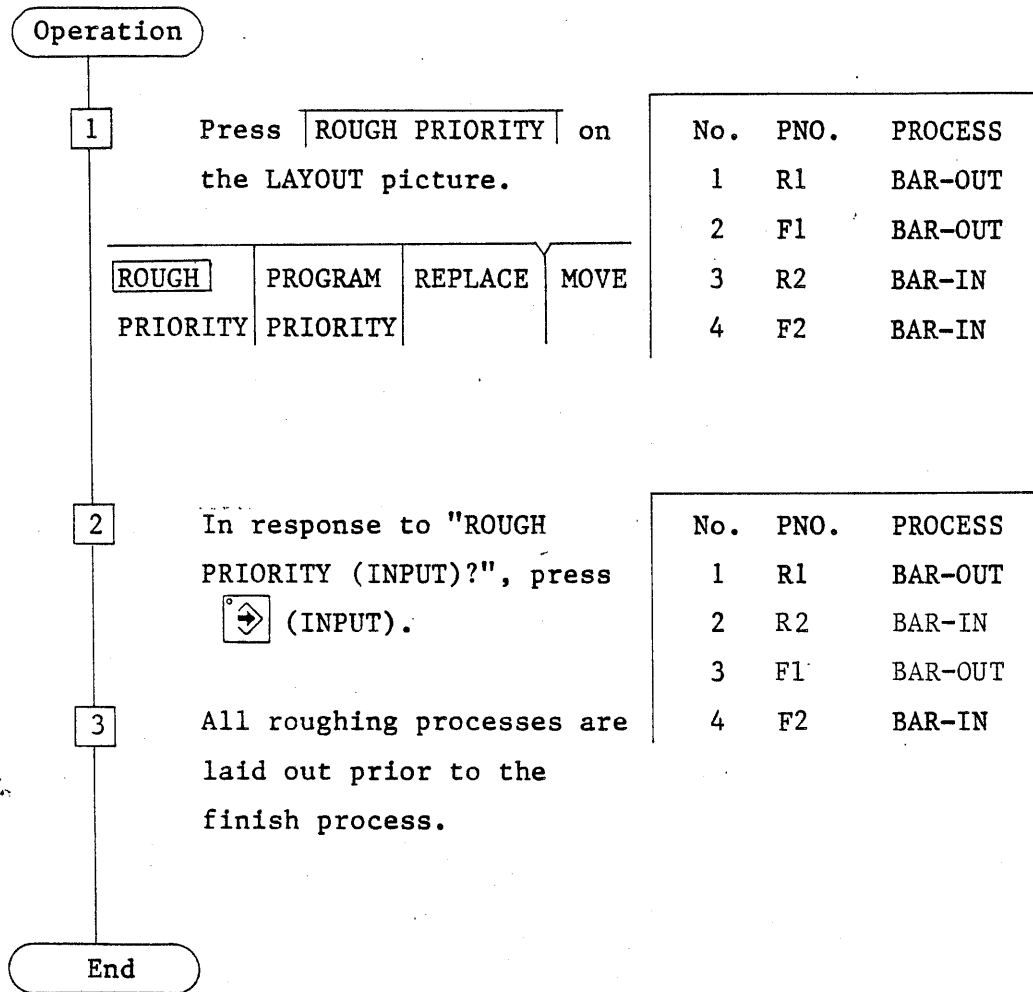


2 ROUGH PRIORITY

Once RT (roughing tool) and FT (finishing tool) have been entered in the MAZATROL Program, roughing and finish machining are alternately performed in the PROGRAM PRIORITY layout mode.

To carry out all roughing prior to the finish process, it is necessary to lay out this ROUGH PRIORITY function.

To layout in ROUGH PRIORITY mode, carry out the following operation:



Note: Milling rough machining is to be laid out after finish machining.

3 Editing Processes in Order

To partially change the sequence of machining processes after preparing a MAZATROL Program, it is possible to edit processes on the LAYOUT picture.

Editing is performed in either of the following two methods:

- (1): REPLACE Operation Two processes are replaced to each other.
- (2): MOVE Operation One process is moved and continued.

inserted. These operations should be performed in accordance with their respective procedures specified below.

(1) REPLACE

Operation

- 1 Move the cursor to the process number to be replaced.
- 2 Select

REPLACE

 .

PROGRAM
PRIORITY

REPLACE

MOVE

ROUGH
R
- 3 In response to "PROCESS TO CHANGE?", press either

ROUGH R

 for roughing or

FINISH F

 for finishing. In addition, use ten keys to enter the process number.


4 Example

No.	PNO.	PROCESS
1	R 1	BAR-OUT
<u>2</u>	R 2	BAR-IN
3	F 3	THR-OUT
4	F 4	GRV-OUT

To replace No. 2 with No. 4, move the cursor to No. 2 and press

FINISH F

4

 and  in succession.

No.	PNO.	PROCESS
1	R 1	BAR-OUT
2	F 4	GRV-OUT
3	F 3	THR-OUT
4	R 2	BAR-IN

End

(2) MOVE


Operation


1 Move the cursor to the number to be moved and inserted.

2 Select MOVE.

REPLACE	<u>MOVE</u>	ROUGH R
---------	-------------	------------

3 In response to "PROCESS NO. TO INSERT?", press either ROUGH R for roughing and FINISH F for finishing. In addition, use ten keys to enter the process number to be moved and inserted.

4 Example To insert and move No. 4 prior to No.2, press FINISH F, 4 and .

No.	PNO.	PROCESS
1	R 1	BAR-OUT
 2	R 2	BAR-IN
3	F 3	THR-OUT
4	F 4	GRV-OUT

No.	PNO	PROCESS
1	R 1	BAR-OUT
2	F 4	GRV-OUT
3	R 2	BAR-IN
4	F 3	THR-OUT

End

- Note: (1) After correcting a program, alarm 422 "LAYOUT NOT COMPLETE?" may result. In this case, carry out the LAYOUT operation.
- (2) The automatic operation under the MAZATROL Program is performed in accordance with the process sequence entered through the LAYOUT picture.

6-4 Automatic Intersect Point Computation Function

The unknown value of the intersect point can be computed using the automatic intersect point computation function when the starting and final points of the convex or concave arc are unknown but the center of the arc and the radius are known, or when either the starting or final point is unknown but the angle θ is known in the sequence of taper or GRV.

6-4-1 Intersect point computation for convex and concave arc

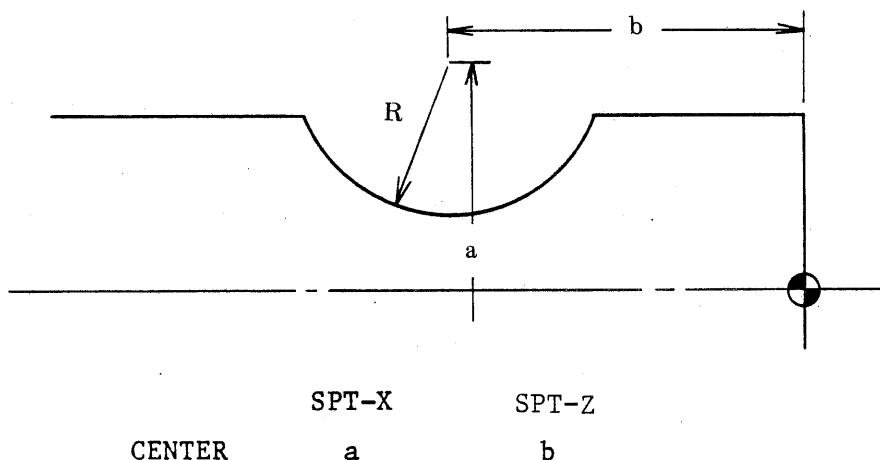
SPT-X	SPT-Z	FPT-X	FPT-Z
?			
	?		
		?	
			?
?		?	
?			?
	?	?	
	?		?

☐ Blank column indicates known data.

On the line below the convex or concave arc sequence in which "?" has been set, select the CENTER on the menu. If the CENTER is pushed, the data on SPT-X and SPT-Z go purple and the following messages are given:

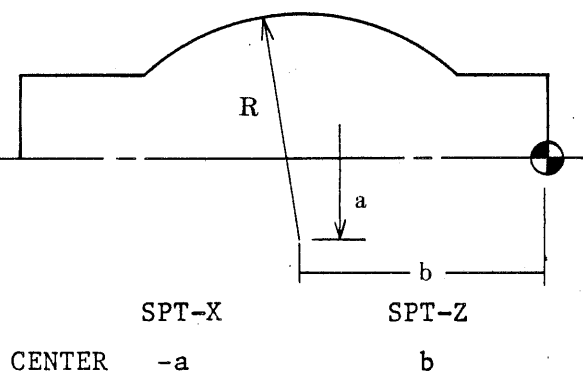
"ARC CENTER-X?" "ARC CENTER-Z?"

Set the X coordinate (a) and Z coordinate (b) of the center of the arc.

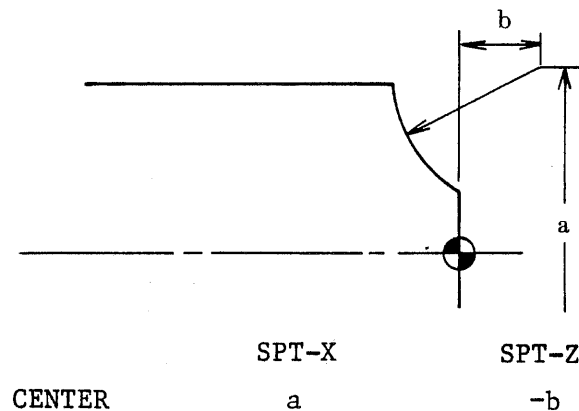


If the center of the arc is below the centerline of the spindle, enter a negative value for X coordinate (a). If the center of the arc is to the right of the program zero point, enter a negative value for Z coordinate (b).

Center of arc is below spindle centerline:



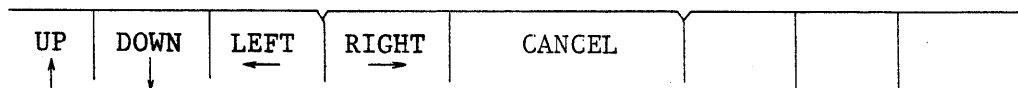
Center of arc is to the right of the program zero point:



When the cursor is put to the ROUGH R, the following message is given:

"INTERSEC POS OF START POINT?" "INTERSEC POS OF FINAL POINT?".

Select the intersect point position of the starting and final points from the menu.

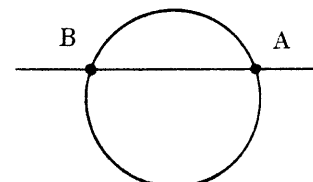


If a straight line is drawn through a circle, it will intersect with the circumference at two points unless the straight line is a tangent.

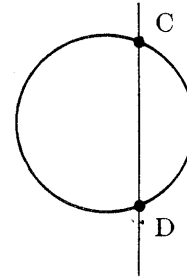
The intersect point position selection means which intersect point of the two obtained for the given circle should be used.

Intersect point position selection:

- o If the X-coordinate is known:
Select RIGHT
→ to use the point A as intersect point and select LEFT
← to use the point B as intersect point.



- o If the Z-coordinate is known:
Select **UP↑** to use the point C
as intersect point and select
DOWN↓ to use the point D as
intersect point.



Starting point, final point and intersect point position:

- o If the starting point is unknown: Select the intersect position of the starting point only.
- o If the final point is unknown: Select the intersect position of the final point only.
- o If both starting and final points are unknown: Select the intersect positions of both starting and final points.

6-4-2 Intersect point computation for taper, GRV sequence

Computable combination when "?" is set:

SPT-X	SPT-Z	FPT-X	FPT-Z
?			
	?		
		?	
			?

☐ Blank column indicates known data.

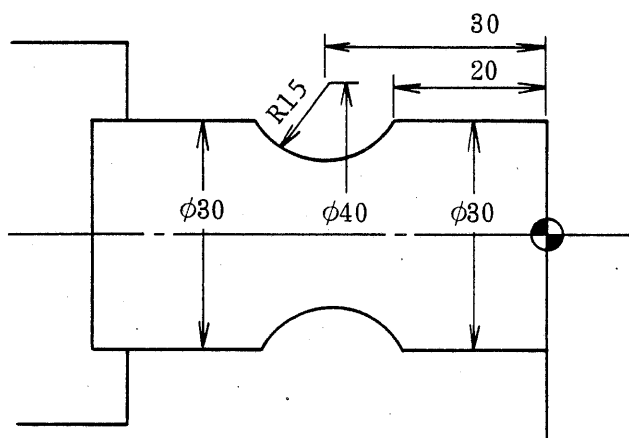
- o Refer to the description of taper in 3-3-2 for the setting of angle θ . Similarly, refer to the GRV sequence data.


When the intersect point positions of the starting and final points have been set, change the menu and select the CALCULAT or FIGURE CHECK from the menu.

If the CALCULAT is selected, automatic calculations are conducted and the calculated value is entered in the position of "?".

If the FIGURE CHECK is selected, the graphics picture appears and the shape formed by including the automatically calculated value is drawn. If the program is called again, the value calculated automatically is entered.

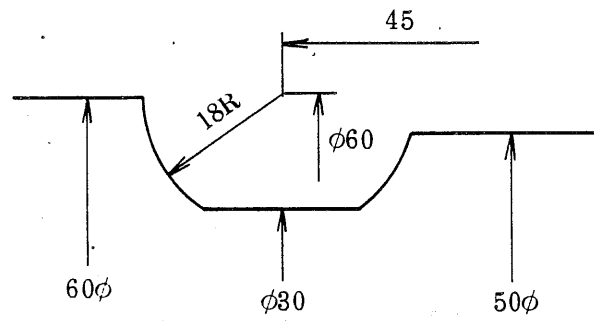
Program example 1:





SEQ	SHP	S-CNR	SPT-X	SPT-Z	FPT-X	FPT-Z	F-CNR	RADIUS	SRF	\$	M
1	LIN	C0			30	20					
2		C0	30	20	30	?	C0	15			
3	CTR		40	30							

If automatic calculation are made, "44.142" is put in "?".

Program example 2:

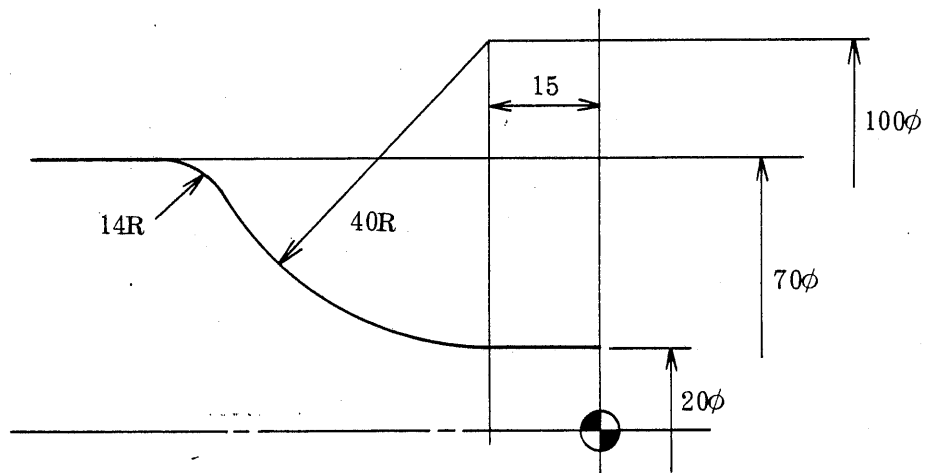


SHP	S-CNR	SPT-X	SPT-Z	FPT-X	FPT-Z	F-CNR	RADIUS	SRF	\$
	C0	50	?(Z ₁)	30	?(Z ₂)	C0	18		
CTR		60	45					→	→
	C0	30	?(Z ₃)	60	?(Z ₄)	C0	18		
CTR		60	45					←	←

The following are found from automatic calculations:

$$\begin{aligned}
 Z_1 &= 27.708 & Z_2 &= 35.05 \\
 Z_3 &= 54.95 & Z_4 &= 63
 \end{aligned}$$

Program example 3:



SHP	S-CNR	SPT-X	SPT-Z	FPT-X	FPT-Z	F-CNR	RADIUS	SRF	\$
☐	C0	20	15	70	?	R14	40		
CTR		100	15						←

"52.081" is put in "?" after automatic calculations.

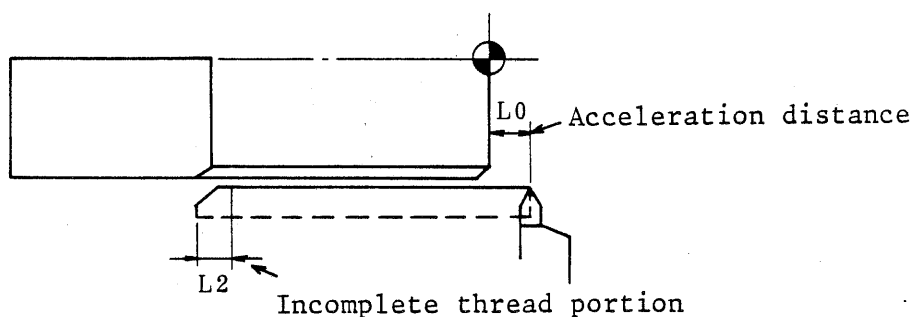
Note : In some cases, an error occurs at the third lower digit than the decimal point due to the automatic intersect point calculation.

7. APPENDIX

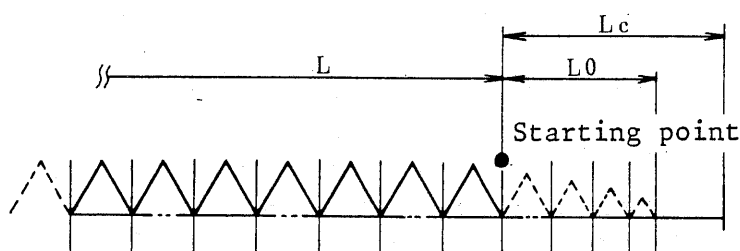
7-1 Various Thread Formulas and Principles

(1) Incomplete thread portion

Since threading feedrate is automatically increased and decreased, the pitches will not be uniform near to the starting and final points. In programming, therefore, it is necessary to provide allowances for the acceleration distance, L_0 , and the imperfect thread length L_2 .



Threading acceleration distance (approach distance)



where, L : Effective thread portion
 L_0 : Acceleration distance
 L_c : Acceleration distance clamp value

To avoid a pitch error due to a subsequent delay in the acceleration of the servo motor, threading must be started at a point displaced from the starting point by the acceleration distance (L_0) obtained from the following expression:

$$L_0 = \frac{F}{60} \cdot \frac{1}{KP} (-\ln a - 1 + a) \quad (\text{mm})$$

where, F : Threading speed (mm/min)

KP : Position loop gain (sec⁻¹)

a : Allowable pitch error rate (%)

ln a : Logarithm the base of which is e
(equivalent to log_e a)

Threading speed (F) = spindle rpm x thread pitch

Position loop gain (KP) = Parameter LP: threading
acceleration distance calculating constant, with l = 1 sec.⁻¹

Allowable pitch error rate (a) = Parameter A2: threading
acceleration pitch error rate, with l = 0.1%

$$\text{Pitch error} = \frac{\text{A2}}{1000} \times \text{lead}$$

To avoid an unnecessarily large acceleration distance (L₀),
the maximum value (clamp value) should be entered as the
parameter.

Parameter U35: Acceleration distance clamp value (L_c) with
l = 0.1 x lead

(a) calculated value < parameter value entered o

(b) calculated value > parameter setting value x

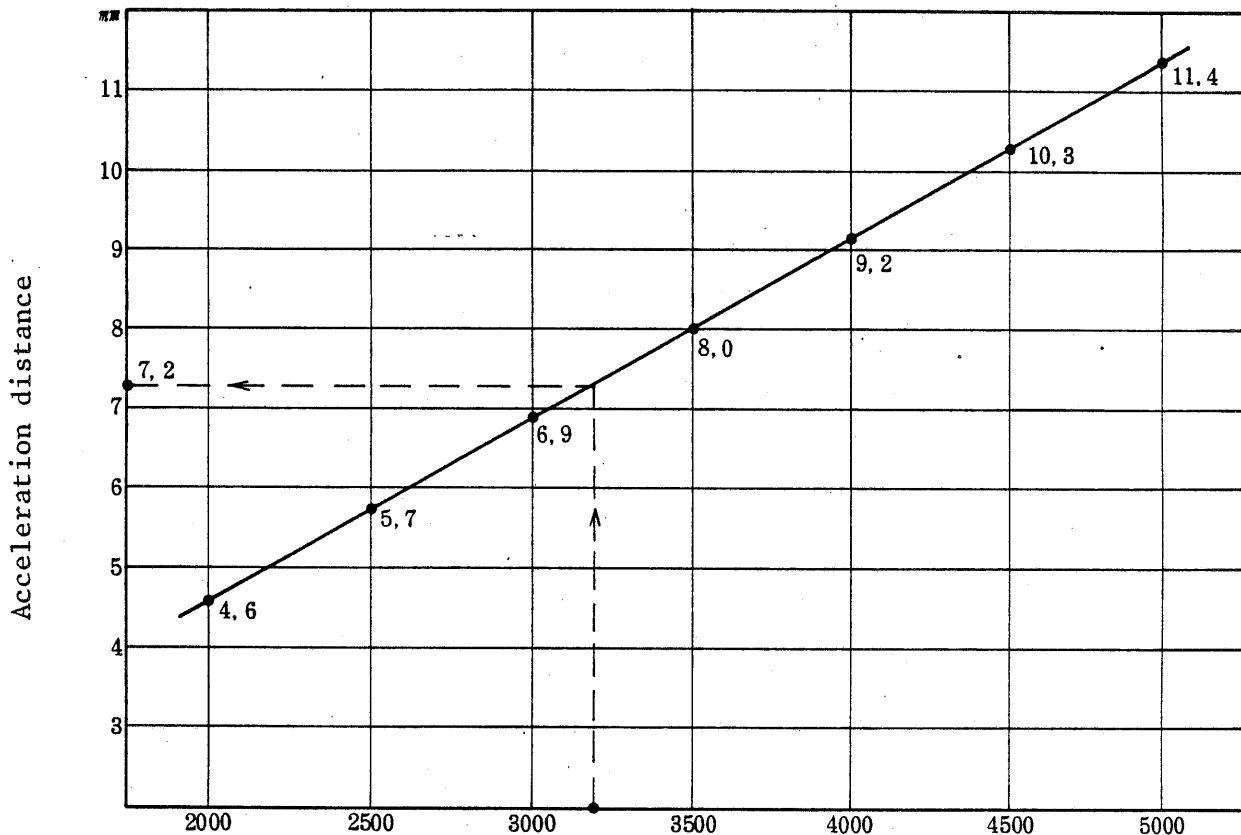
If parameter U21 = 1 in case (b), this will result in alarm
532 "ACCELERATION DISTANCE EXCEED".

In this case, it is necessary to:

1. Decrease the threading speed (F),
2. Increase the allowable pitch error rate (a) and
3. Increase the maximum value (clamp value).

Simple method of obtaining the threading acceleration distance:

During threading, an acceleration distance is set automatically. If the distance set is found inappropriate, it will be necessary to alter Parameter U35.



Threading acceleration: $F = R \times P$ (mm/min.)

(Example)

An M30 screw with $P = 2$ at surface speed $V = 120$ m/min. is to be machined. What acceleration distance should apply?

$$\text{Spindle speed } N = \frac{1000 V}{\pi D} = \frac{120000}{94.2478} = 1273 \text{ rpm.}$$

$$\text{Threading rate } F = R \times P = 1273 \times 2.5 \div 3182 \text{ mm/min}$$

Therefore, the acceleration distance is approx. 7.2 mm

Surface speed parameters

$$\text{A2} = 20 \quad \text{LP} = 33$$

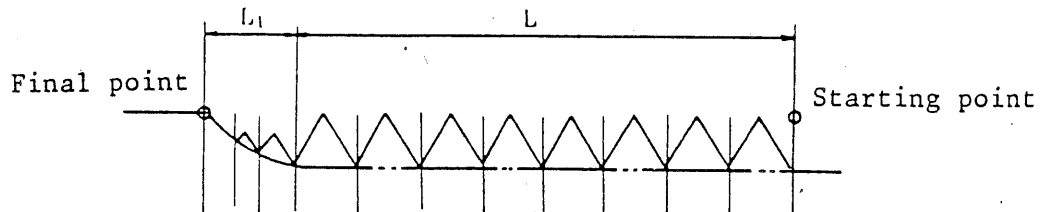
$$\text{TCC} = 20$$

R : Spindle rpm

P : Pitch

Incomplete thread portion in end finishing

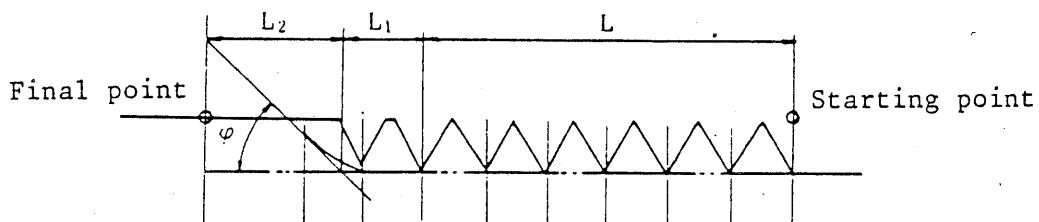
a) Thread end finishing method 1 (without chamfering)



where, L : Effective thread portion

L_1 : Incomplete thread portion with unequal pitches (due to delay)

b) Thread end finishing method 2 (with chamfering)



where, L : Effective thread portion

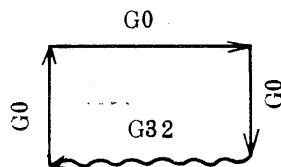
L_1 : Incomplete thread portion with an equal pitch (due to delay)

L_2 : Incomplete thread portion with an equal Pitch (chamfering stroke: parameter **U36**)

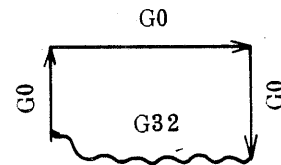
ψ : Chamfering angle

- o **[A1]** Thread end finishing constant changeover selection distance with **[1 = 0.001 mm]**, **[standard: 0]**

During thread end finishing a directional movement is at the rapid feedrate. Only when a distance of this move is smaller than the value entered in **[A1]**, will the rising time constant be changed from the rapid feed time constant to the machining feed time constant.



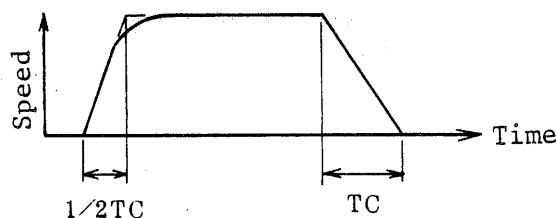
Without chamfering



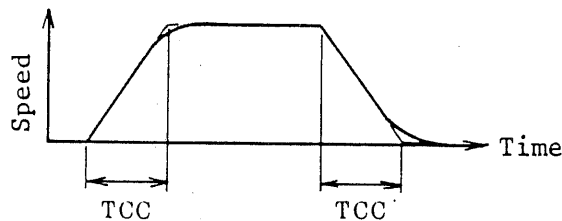
With chamfering

Cutting feed time constant **[TCC]** 30 msec is smaller than rapid feed time constant **[TC]** 120 msec. Once the time constant has been changed over, therefore, the thread end finishing feedrate will rise sooner. Nevertheless, the incomplete thread portion length does not change. In other words, TPM is valid when the distance of G0 in the illustration above is smaller than the value set in **[A1]**. Setting **[0]** will result in a rapid feed time constant.

- o **[TC]** Rapid feed accelration/deceleration time constant
Standard : X.Z : 120



- o **TCC** Cutting feed acceleration/deceleration time constant
Standard : 30



- a) Cutting feed acceleration/deceleration time constant without chamfering:

This time constant may be obtained from the below expression. For rough values, see Table 1.

$$\left(\frac{1}{K_s} + \frac{1}{K_p} \right) \times \frac{F \times 1000}{60} = L_2 \quad \frac{1}{K_s} = 0.03 \text{ sec} \quad \frac{1}{K_p} = \frac{1}{33} = 0.04 \text{ sec}$$

$$L_2 = 0.07 \times \frac{F \times 1000}{60} \text{ mm} \quad \frac{1}{K_s} = \text{[TCC]} \quad F : \text{m/min}$$

$$K_p = 25$$

- b) Cutting feed acceleration/deceleration time constant with chamfering:

If Chamfering Width **[CH]** $\times 0.1 = C$, then

the incomplete thread portion = $C + L_2$

(Example 1) When threading an M50 screw at $V = 100$ and $P = 2.0$,

$$F = \frac{1000V}{\pi D} \times 2.0 = \frac{1000 \times 100 \times 2}{3.14 \times 50} = 1273 \text{ mm/min}$$

From **Table 1**, $L_2 = 15 \text{ mm}$ may be obtained.

(Example 2) When threading an M30 screw of JIS 2nd class at $V = 100$ and $P = 2.5$:

Based on Table 2, Parameter **[A2]** should be set to approximately **[13]**.

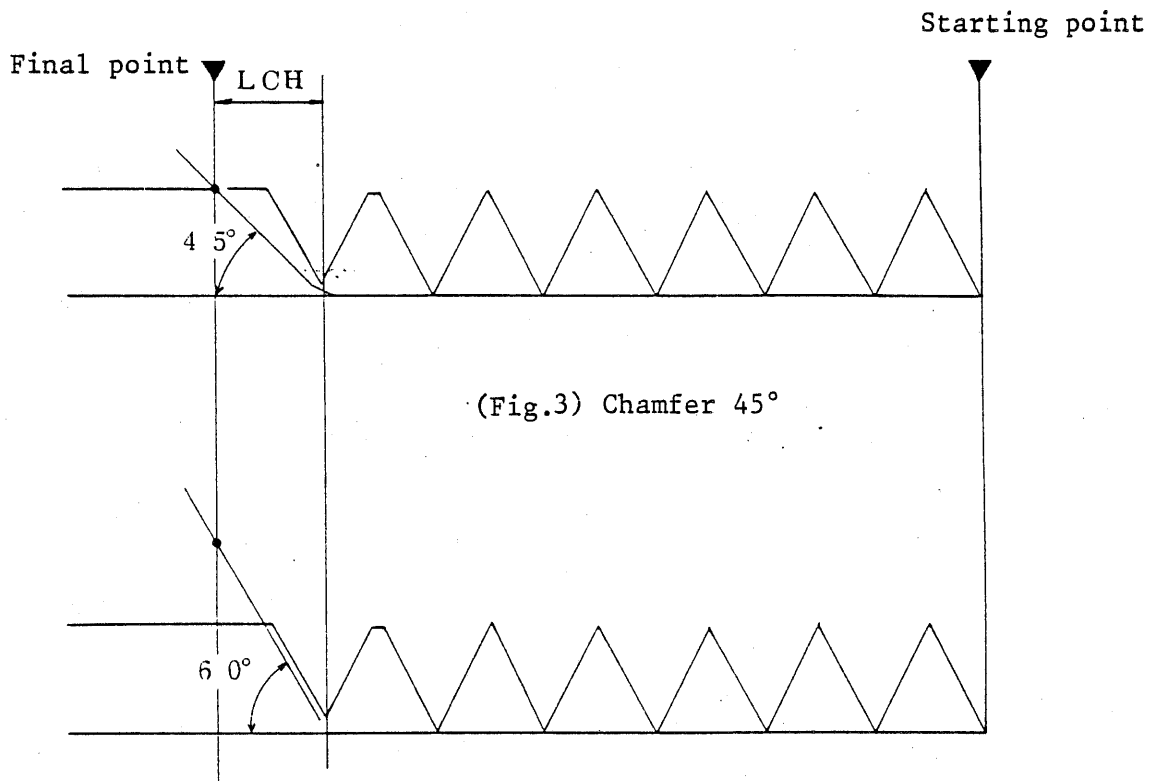
$$F = \frac{1000V}{\pi D} \times 2.5 = 2652 \text{ mm/min}$$

From **Table 2**, $L_0 = 7 \text{ mm}$ is obtained.

It is necessary, therefore, to set **[U35]** to $7/2.5 \times 10 = 28$ or more.

(2) Chamfering effects

- a) Difference of 45° and 60° chamfering at an equal chamfering distance (with a following delay ignored for convenience)



(Fig.4) Chamfer 60°

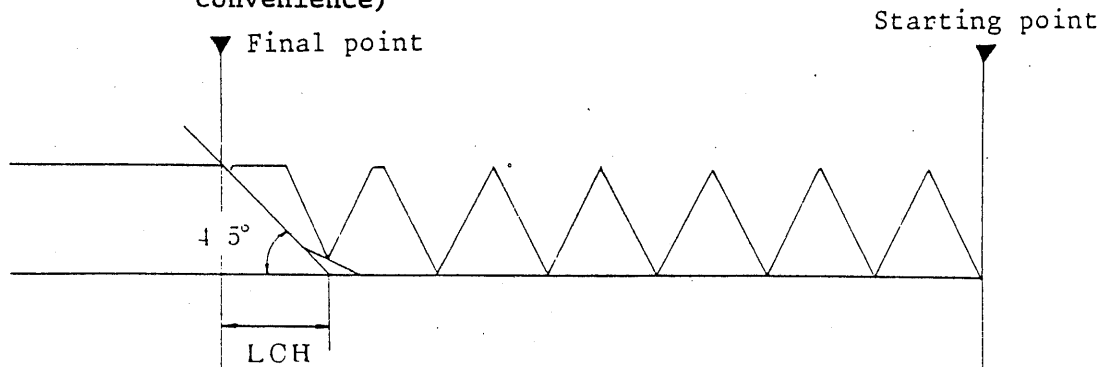
Chamfering distance (LCH) is given as parameter U36

Figs. 3 and 4 above show the difference between Chamfer 1 (45°) and Chamfer 2 (60°) with the chamfering distance kept the same.

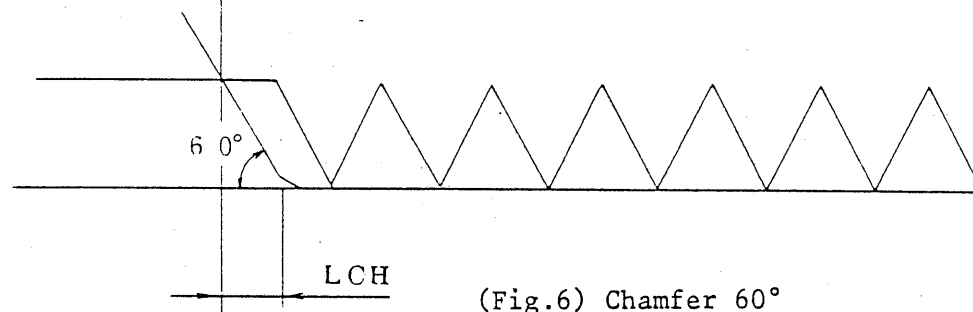
Chamfering at 60° will result in a larger thread variation without allowing a thread gage to easily enter into the innermost cutting thread location. (The appearance, however, is acceptable.)

To use Chamfer 45° and Chamfer 60° properly, therefore, it is necessary to adjust the chamfering distance (LCH) by changing the value of Parameter U36

- b) Difference between 45° and 60° , with chamfering distance adjusted (with the following delay ignored for convenience)



(Fig. 5) Chamfer 45°



(Fig. 6) Chamfer 60°

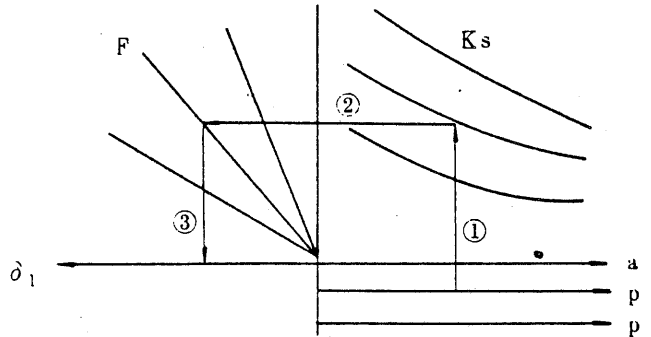
Figs. 5 and 6 show the difference between Chamfer 45° and Chamfer 60° , with an appropriate chamfering distance (LCH). The figures show a larger effective thread portion is available at Chamfer 60° so that a thread gage can be easily inserted to the innermost cutting thread location.

When the requirement "chamfering Distance > Following Delay" is satisfied and if an appropriate chamfering distance is given, a larger portion will be effectively threaded at Chamfer 60° . As a result, a thread gage may be entered to the innermost cutting thread location.

Repeated selection of Chamfer 45° and Chamfer 60° is disadvantageous. After learning their respective merits, determine which chamfer be applied, 45° or 60° . It is also desirable to give a more suitable chamfering distance beforehand.

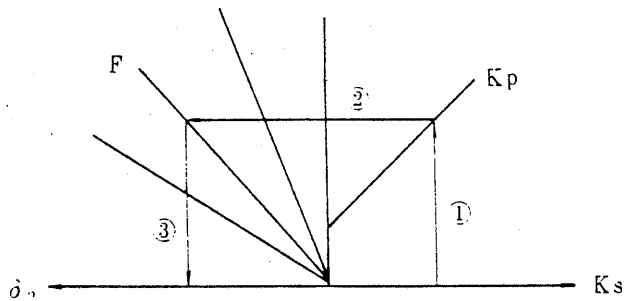
(3) Graphs of incomplete thread portions (How to read a graph)

a) Acceleration distance graph



Once a screw class and a pitch (P) have been determined, Line ① starting from a division on the p-axis intersects a curve of smoothing circuit gain (K_s). Next, Line ② intersects the threading speed (F). Furthermore, Line ③ intersects the U-axis, where an acceleration distance can be obtained.

b) Incomplete thread portion



With the smoothing circuit gain (K_s) determined, draw a line ① perpendicular to K_s axis to obtain a point of intersection between the line and the one representing the position loop gain (K_p). After that draw a line ② parallel to the K_s axis until it crosses the line representing the employed thread cutting speed (F). From this point of intersection, draw a line ③ perpendicular to V axis toward that axis. The value read on the V axis is the incomplete thread length (δ_2) to be obtained. When chamfering is done at an angle 45 deg., add one pitch to the value obtained above.

Table 1

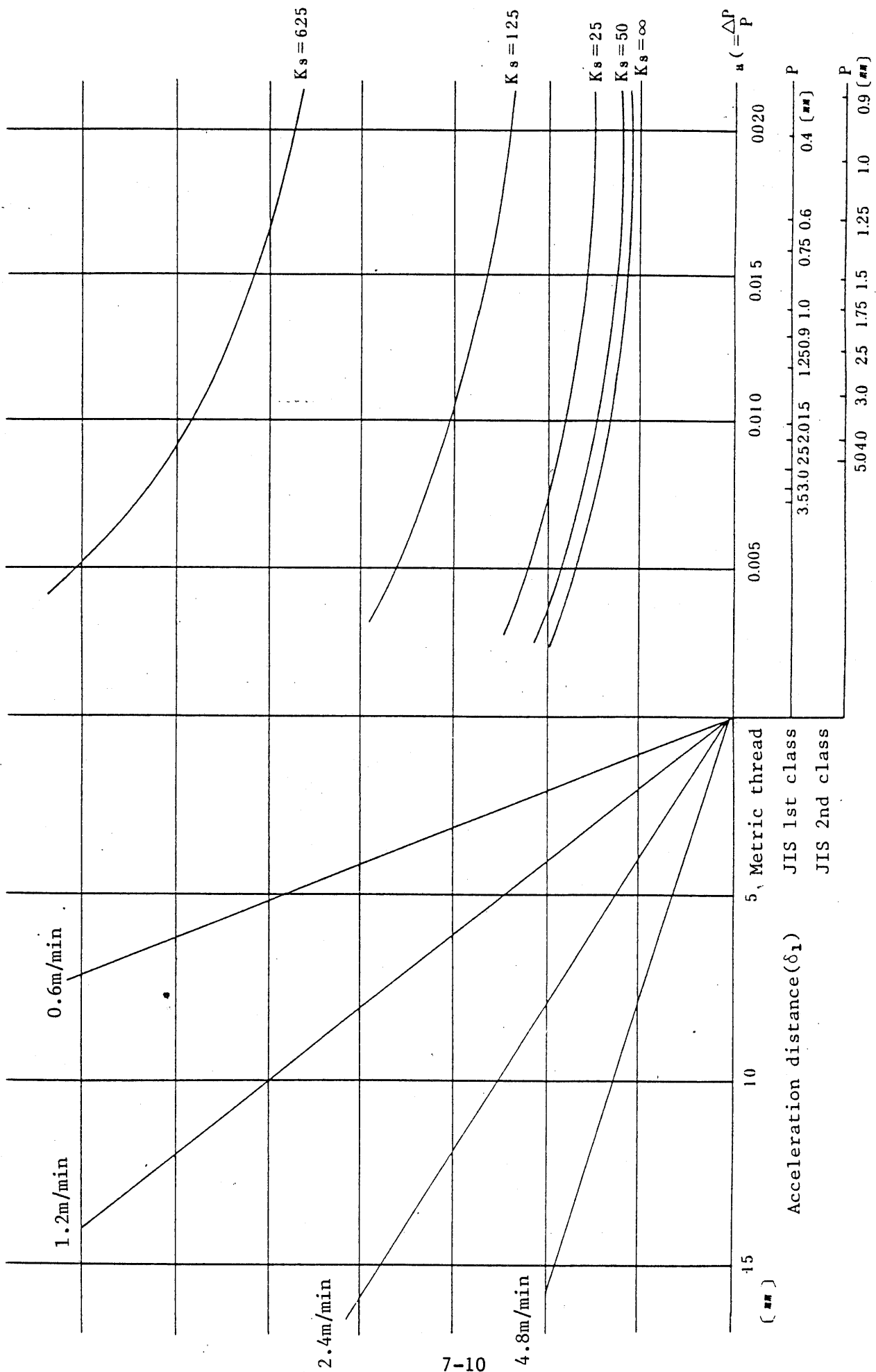
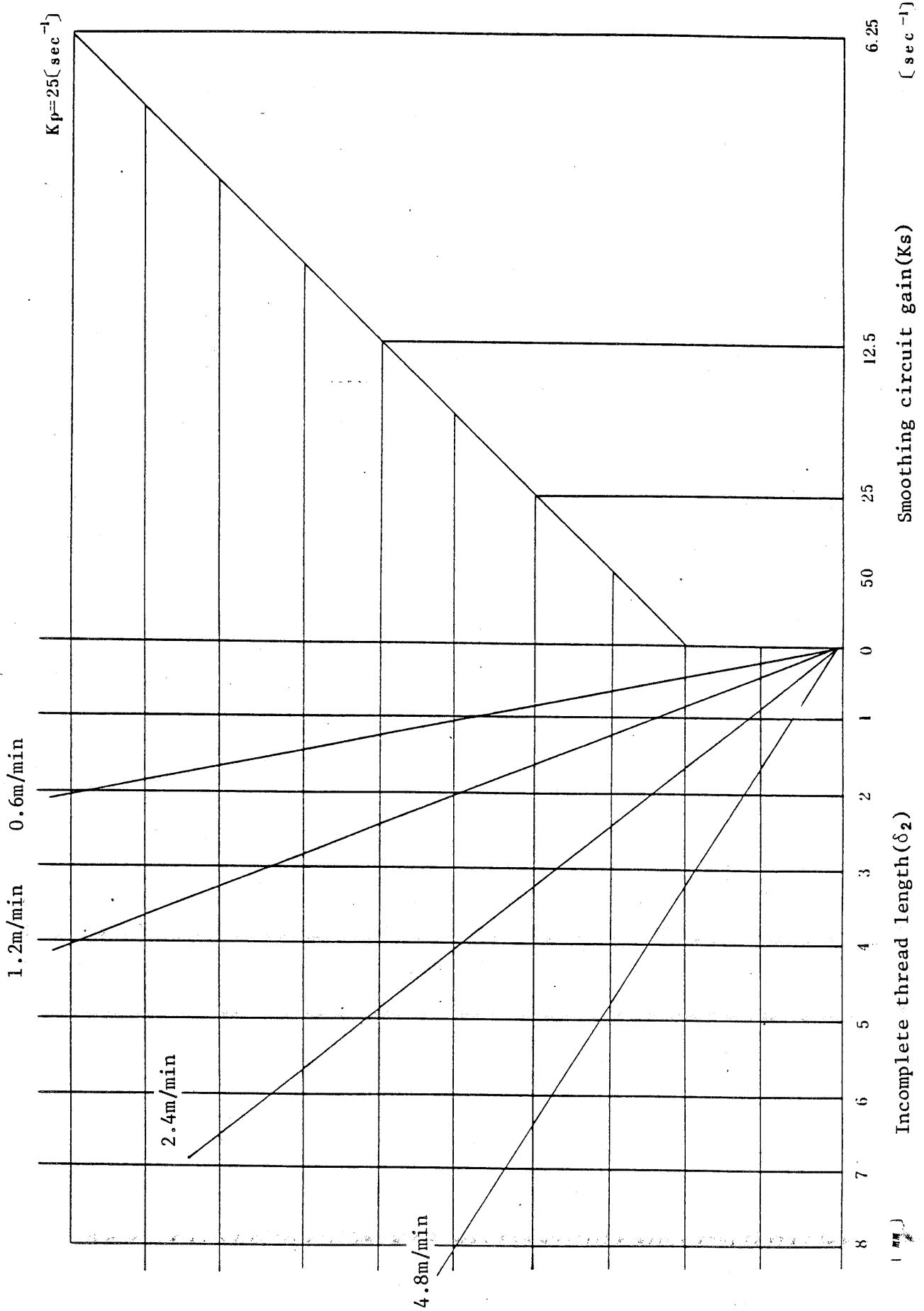


Table 2



(4) Automatic determining a thread height

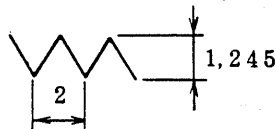
A height of thread is automatically determined.

Nevertheless, it is a variable parameter.

(Parameter)	<u>U37</u> (O.D., Face)	
(mm)	<u>U38</u> (I.D.)	: 6495 (thread angle 60°) (%/100)
	<u>U39</u> (O.D., Face)	
(inch)	<u>U40</u> (I.D.)	: 6403 (thread angle 55°) (%/100)
(Standard value entered upon shipment)		

(Example)

At P = 2.0, the standard thread height is as follows:



If a thread height of 1.3 mm is needed, use the following expression:

$$\begin{aligned} & \frac{1.3}{1.245} \times 0.6495 = \\ & \text{(height of thread to be changed)} : \text{(standard height)} = \\ & \text{(constant to be changed)} : \text{(standard value)} \end{aligned}$$

$$X = \frac{1.3}{1.245} \times 0.6495 = 0.6782$$

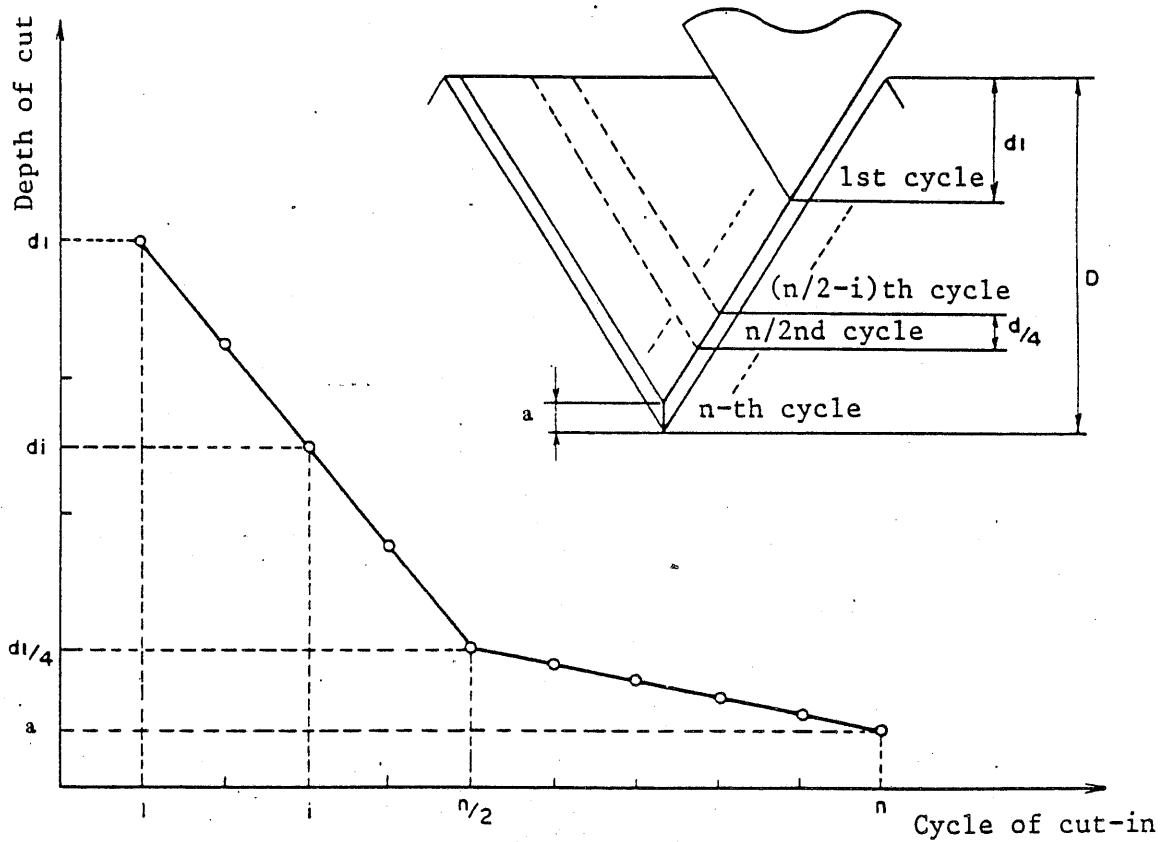
Therefore, the parameter to be changed should be set to :

U37 : 6782.

Later, a thread height by size will automatically be determined at a value increased by 4.4% of the standard.

$$6782/6495 = 1.044$$

(5) Depth of cut and cycles of cut-in in threading



i) With $1 \leq i \leq \frac{n}{2}$ $d_i = d_1 - \frac{3}{2} \frac{i-1}{n-2} d_1$

ii) With $\frac{n}{2} \leq i \leq n$ $d_i = (\frac{d_1}{2} - a) - (\frac{d_1}{2} - 2a) \frac{i}{n}$

where, D : Total depth of cut

d_1 : Depth of cut in 1st cycle

d_i : Depth of cut in i -th cycle

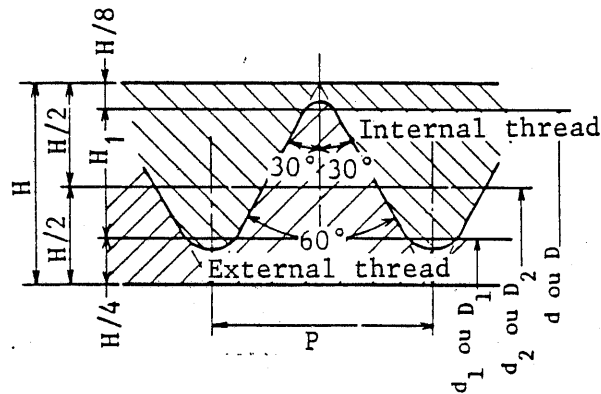
a : Depth of cut in final cycle

(parameter $\frac{U33}{2}$)

n : Cycles of cut-in ($3 \leq n \leq 99$)

With $\sum_{i=1}^n d_i = D$, $\frac{D - \sum_{i=1}^n d_i}{n}$ is added to d_k above.

- (6) Reference thread shape, depth of cut and number of cutting cycles for coarse metric thread
(Reference metric thread shape)



$$H = 0.866025p$$

$$H_1 = 0.541266p$$

$$d_2 = d - 0.649519p$$

$$d_1 = d - 1.082532p$$

$$D = d \quad D_2 = d_2 \quad D_1 = d_1$$

- 1) Threading depth of cut and number of cutting cycles
Example of machining with S45 cemented carbide cutting tool (for external thread)

Cutting feedrate

Pitch	P	1,00	1,25	1,50	1,75	2,00	2,60	3,00	3,50	4,00	4,50	5,00	5,50	6,00
Height	H2	0,60	0,74	0,89	1,05	1,19	1,49	1,79	2,08	2,38	2,68	2,98	3,27	3,57
	H1	0,541	0,677	0,812	0,947	1,083	1,353	1,642	1,894	2,165	2,435	2,706	2,977	3,248
	R	0,07	0,09	0,11	0,13	0,14	0,18	0,22	0,25	0,29	0,32	0,36	0,40	0,43
number of cutting cycles	1	0,25	0,35	0,35	0,35	0,35	0,40	0,40	0,40	0,40	0,40	0,45	0,45	0,45
	2	0,20	0,19	0,20	0,25	0,25	0,30	0,35	0,35	0,35	0,35	0,35	0,40	0,40
	3	0,10	0,10	0,14	0,15	0,19	0,22	0,27	0,30	0,30	0,30	0,30	0,35	0,35
	4	0,05	0,05	0,10	0,10	0,12	0,20	0,20	0,25	0,25	0,30	0,30	0,30	0,30
	5		0,05	0,05	0,10	0,10	0,15	0,20	0,20	0,25	0,25	0,25	0,30	0,30
	6			0,05	0,05	0,08	0,10	0,13	0,14	0,20	0,20	0,25	0,25	0,25
	7				0,05	0,05	0,05	0,10	0,10	0,15	0,20	0,20	0,20	0,25
	8					0,05	0,05	0,05	0,10	0,14	0,15	0,15	0,15	0,20
	9						0,02	0,05	0,10	0,10	0,10	0,15	0,15	0,15
	10							0,02	0,05	0,10	0,10	0,10	0,10	0,15
	11							0,02	0,05	0,05	0,10	0,10	0,10	0,10
	12								0,02	0,05	0,09	0,10	0,10	0,10
	13								0,02	0,02	0,05	0,09	0,10	0,10
	14									0,02	0,05	0,05	0,08	0,10
	15									0	0,02	0,05	0,05	0,08
	16										0,02	0,05	0,05	0,05
	17										0	0,02	0,05	0,05
	18											0,02	0,05	0,05
	19											0	0,02	0,05
	20												0,02	0,05
	21												0	0,02
	22													0,02
	23													0
	24													
	25													
	26													

*** Numerical values above are given for reference. They vary with chip conditions etc.

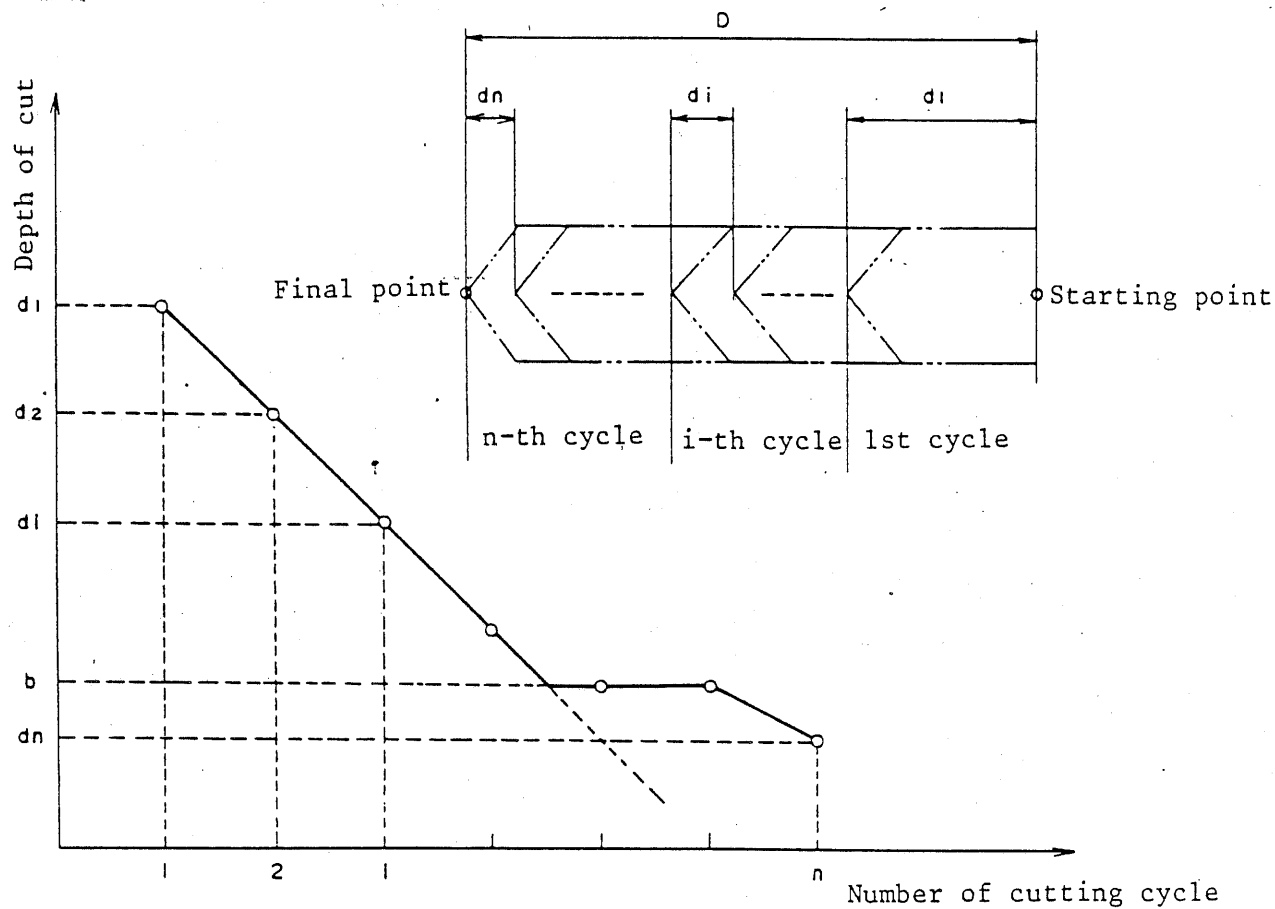
- 2) Internal thread lower hole diameter (Data given below are for reference. For details, see any mechanical engineering handbook.)

Thread				Lower hole diameter										Reference			
External thread	O.D. (d)	Pitch (p)	Height of engagement (H ₁)	Series (engagement rate) (%)										Thread I.D.			
														Min.	Max. Size		
				100	95	90	85	80	75	70	65	60	Size	Class 1	Class 2	Class 3	
M20	20,000	2,5	1,353	17,3	17,4	17,6	17,7	17,9	18,0	18,1	18,2	18,4	17,294	17,649	17,744	17,854	
M22	22,000	2,5	1,353	19,3	19,4	19,6	19,7	19,9	20,0	20,1	20,2	20,4	19,294	19,649	19,744	19,854	
M24	24,000	3	1,624	20,8	20,9	21,1	21,2	21,4	21,6	21,7	21,9	22,1	20,752	21,152	21,252	21,382	
M27	27,000	3	1,624	23,8	23,9	24,1	24,2	24,4	24,6	24,7	24,9	25,1	23,752	24,152	24,252	24,382	
M30	30,000	3,5	1,894	*	26,4	26,6	26,8	27,0	27,2	27,1	27,5	27,7	26,211	26,661	26,771	26,921	
M33	33,000	3,5	1,894	*	29,4	29,6	29,8	30,0	30,2	30,4	30,5	30,7	29,211	29,661	29,771	29,921	
M36	36,000	4	2,165	31,7	31,9	32,1	32,3	32,5	32,8	33,0	33,2	33,4	31,670	32,145	32,270	32,420	
M39	39,000	4	2,165	34,7	34,4	35,1	35,3	35,5	35,8	36,0	36,2	36,4	34,670	35,145	35,270	35,420	
M42	42,000	4,5	2,436	*	37,4	37,6	37,9	38,1	38,4	38,6	38,8	39,1	37,129	37,659	37,799	37,079	
M45	45,000	4,5	2,436	*	40,4	40,6	40,9	41,1	41,4	41,6	41,8	42,1	40,129	40,659	40,799	40,079	
M48	48,000	5	2,706	42,6	42,9	43,1	43,4	43,7	43,9	44,2	44,5	44,8	42,587	42,147	43,297	43,187	

Note 1. $H_1 = 0.841266p$

2. Lower hole diameter = $d - 2 \times H_1 \left(\frac{\text{engagement rate}}{100} \right)$

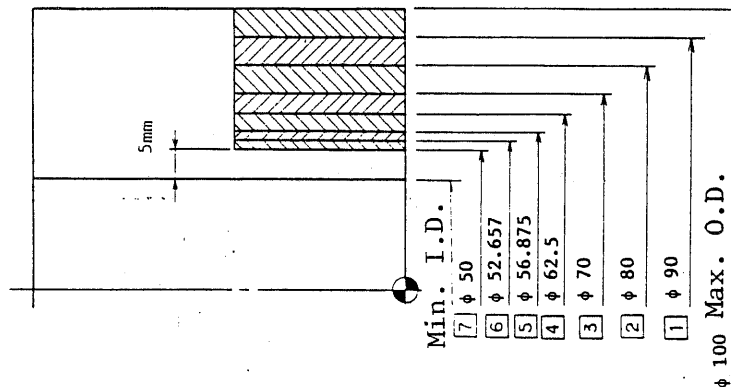
7-2 Graphs Showing the Relations between Depth of Cut and Number of Cutting Cycles for Drilling (DRL)



Cut #1	d_1	Depth of cut
#2	$d_1 - d_2$	Cut attenuation
#3	b	Min. cut depth

7-3 Reducing Depth of Cut

When the workpiece is thinner, the depth of cut is automatically decreased. This is called "Cutting Reduction Function." A reduced rate may be set in **Parameter**.



Setting conditions

: Set a reduced rate in **U59**.

Normally, **100%** should be set.

$$A = T \times \frac{K}{100}$$

o Selected cut depth (D) : 5 mm

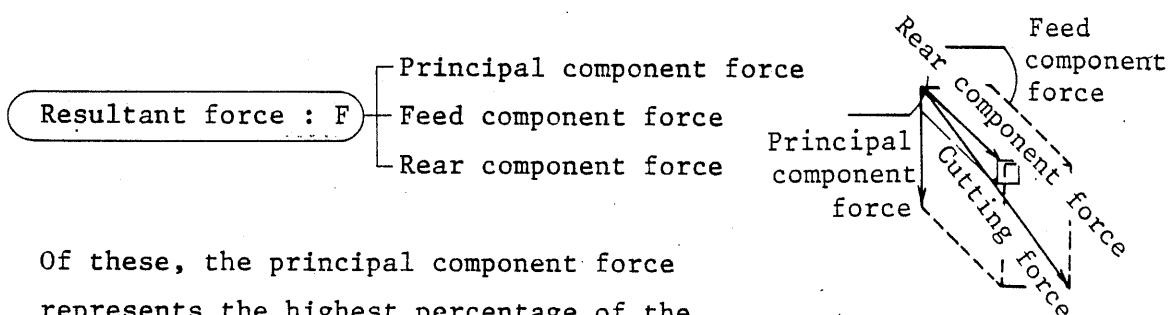
o Reduction rate (K) : 25% To be set as **25**.

Cutting	Residual wall thickness (T)	$T \times K = A$	Cut depth (A ≤ D)	Cutting diameter	Decrease
1st cycle	$\frac{100-40}{2}=30$ (mm)	$30 \times 0.25 = 7.5^*$	5.0 (mm)	φ90	Not decreased
2nd cycle	$\frac{90-40}{2}=25$	$25 \times 0.25 = 6.25^*$	5.0	φ80	Not decreased
3rd cycle	$\frac{80-40}{2}=20$	$20 \times 0.25 = 5.0$	5.0	φ70	Not decreased
4th cycle	$\frac{70-40}{2}=15$	$15 \times 0.25 = 3.75$	3.75	φ62.5	Reduced by T x 25%
5th cycle	$\frac{62.5-40}{2}=11.25$	$11.25 \times 0.25 = 2.812$	2.812	φ56.875	Reduced by T x 25%
6th cycle	$\frac{56.875-40}{2}=8.437$	$8.437 \times 0.25 = 2.109$	2.109	φ52.657	Reduced by T x 25%
7th cycle	$\frac{56.657-40}{2}=6.328$	—	1.328	φ50	Remaining cut depth

$$\text{Residual wall thickness : } T = \frac{\text{O.D. then} - \text{Minimum I.D.}}{2}$$

*** If the calculated cut depth should excess D , D is used as a cut depth.

7-4 Cutting Force and Cutting Power



Of these, the principal component force represents the highest percentage of the resultant force, i.e., 75 - 90%, therefore, it can be safely said that it approximates to the principal component.

$$\text{Cutting Power:HP} = \frac{\text{Depth of cut} \times \text{Feed} \times \text{Surface Speed} \times \text{specific cutting force}}{75 \times 60}$$

$$= \frac{d.F.V.K_s}{75 \times 60}$$

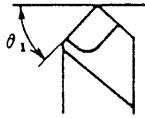
Example: Material SBN STL (tensile strength: 70 kg/cm²) , d=5mm,
F=0.5mm/rev, V=110m/min, K_s=200kg/mm², D=φ250

$$\text{HP} = \frac{5 \times 0.5 \times 110 \times 200}{75 \times 60} = 12.2 \div 13 \text{ horsepower}$$

Value of k_f (kg/mm²)

Material	Tensile strength and hardness	Feedrate mm/rev.				
		0.04	0.1	0.2	0.4	1.0
Carbon steel	40 kg/mm ²	350	290	250	212	173
	60 kg/mm ²	430	356	300	255	212
	80 kg/mm ²	500	410	350	300	245
	100 kg/mm ²	550	450	385	330	270
	140 kg/mm ²	650	530	460	395	320
Alloy steel	180 kg/mm ²	355	700	600	510	420
	120 HB	185	142	118	97	75
	160 HB	260	200	166	137	106
Cast iron	200 HB	340	260	215	178	137

Value of θ_k



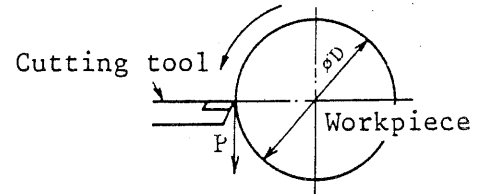
Cutting edge angle θ_1	30°	45°	60°	90°
Work material				
Steel	1.27	1.16	1.09	1.00
Cast iron	1.21	1.13	1.07	
Aluminum	1.15	1.09	1.06	
Brass, Bronze	1.10	1.06	1.04	

Value of r_k

Type of chip Rake angle r	0°	5°	10°	15°	20°	25°	30°	35°	40°
Flow type	1.00	0.96	0.92	0.88	0.84	0.80	0.75	0.71	0.66
Shear or Tear type		0.95	0.90	0.85	0.80	0.75	0.69	0.64	0.59

$$K_s = r_k \cdot \theta_k \cdot k_f$$

$$\text{Cutting torque: } T = \frac{A \times D \times K_s}{2000}$$



Example.

$$T = \frac{5 \times 0.5 \times 250 \times 200}{2000} = 62.5 \text{ kg.m}$$

Also, based on the figure to the right side:

$$T = P \times L = \text{Principal component force} \times \text{Radius of workpiece,}$$

therefore,

$$\text{Torque } T = \text{Cutting area} \times \text{Specific cutting force} \times \text{Radius}$$

$$= d.F.K_s.D/2 = 62.5 \text{ kg.m}$$

7-5 Selecting a Range of Spindle Gear Speeds

Spindle speeds are changed automatically through a range of two to several gears so that an optimum cutting condition may be selected. If the operator wants to learn the cutting condition, therefore it is necessary to obtain required horsepower, required torque and surface speed from the expressions already given.

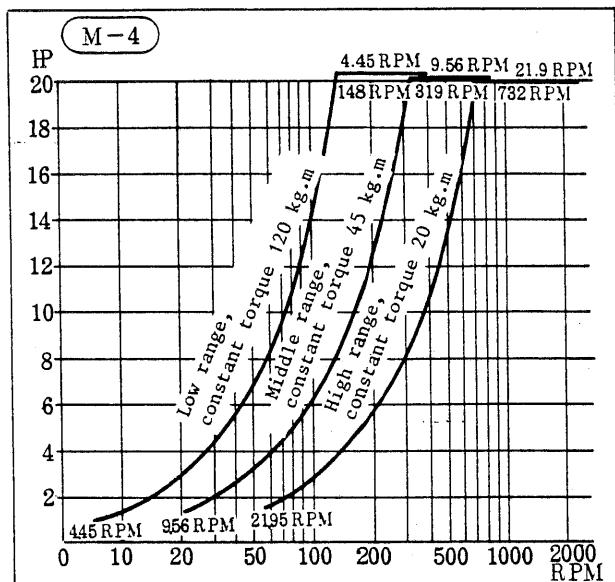
Our catalog contains the diagram of spindle speeds, horsepower and torque illustrated below, to facilitate selecting of the spindle speeds.

The example given in the preceding section can be solved as follows:

$$\phi 250 : 110 = \frac{\pi \cdot D \cdot N}{1000} \quad N = \frac{110 \times 1000}{3.14 \times 250} = 140 \text{ rpm}$$

The range in which 13HP or more is available is the low speed range which provides adequate torque. In the above case, therefore, the low speed range should be selected.

Example



Machining speed =
surface speed

$$V = \frac{\pi \cdot D \cdot N}{1000} \text{ m/min}$$

D: Diameter of material to
be machined (in mm)

N: Speed rpm

Ks: Specific cutting resistance
(roughly estimated value) kg/mm²

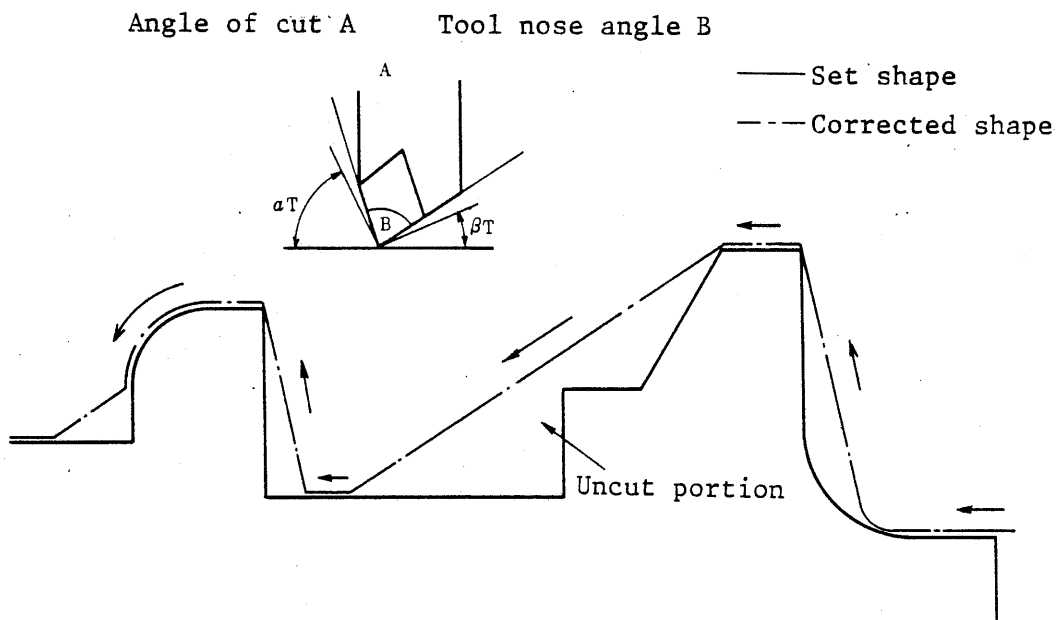
Material	Ks
Mild steel	190
Medium-carbon steel	210
High-carbon steel	240
Low-alloy steel	190
High-alloy steel	245
Cast iron	93
Forged iron	140
Forgeable iron	120
Bronze, Brass	70

7-6 Nose Shape Compensation

(1) Compensating for nose shape

The area in which cutting is possible is restricted by the angle of cut and tool nose angle. A finish shape which has been entered, therefore, is automatically corrected and cut. An uncut portion, therefore, will be left between shapes entered and corrected.

The tool is corrected to move along the path shown below.



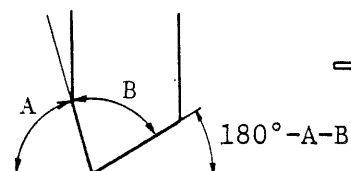
where αT : Tool main cutting edge angle corrected

βT : Tool secondary cutting edge angle corrected

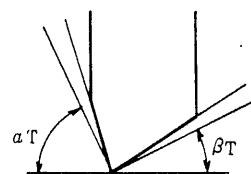
Precautions to be taken in compensating for nose shape:

- 1) Correct, while allowing 3 degrees for the tool cutting edge angle.

Tool cutting
edge angle



Cutting edge
angle corrected



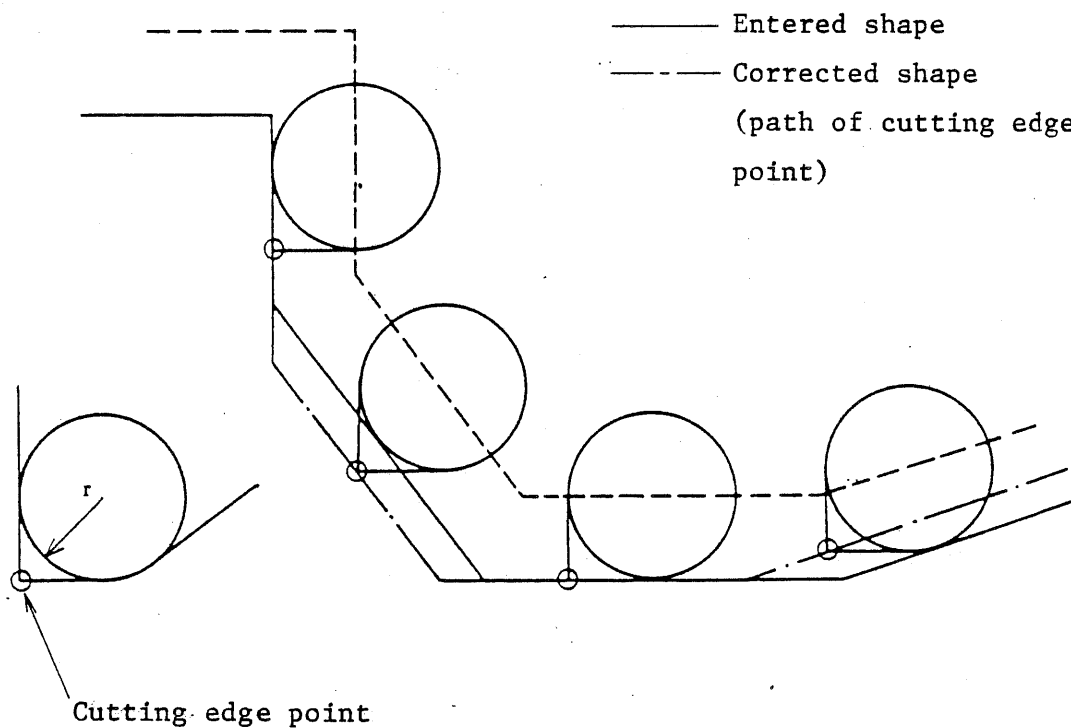
$$\alpha T = A - 3^\circ$$

$$\beta T = 180^\circ - A - B - 3^\circ$$

- 2) The main cutting edge angle is not checked at $\alpha T \geq 90^\circ$.
The secondary cutting edge angle is not checked at $\beta T \leq 27^\circ$.
- 3) If the cutting edge angle differs between rough machining and finishing tools, the main or secondary cutting edge angle whichever may be smaller, should be selected to make the correction.

(2) Nose radius offset

To prevent an error due to the tool nose radius, an entered finishing shape is automatically corrected and cut.

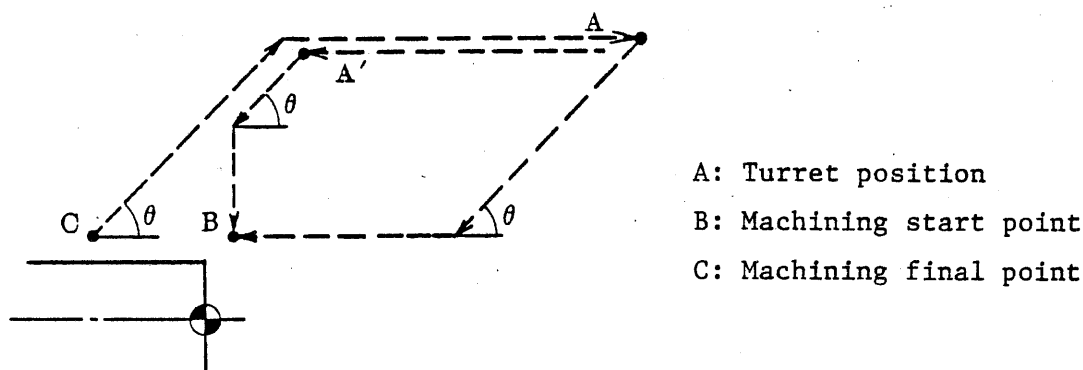


7-7 Simultaneous 2-axis Approach and Return

The movement of the tool to the point where machining is to begin in each process occurs simultaneously along the X and Z axis at the rapid feedrate.

Similarly, the tool is returned in the same way to the specified tool position (fixed point, tool swing position, zero point position, etc.) after machining has ended.

In this manual, the advance to the machining start point and the return to the tool escape position are discussed as simultaneous 2-axis movements; however, the resulting actual tool paths are as shown in the following figure.



The size of the angle θ depends on the rapid feedrates for the X and Z axis.

If these feedrates for both the X and the Z axis are the same, θ will be 45° .

If any interference such as the steady rest is encountered during the two-axis movement to the machining start point or tool escape position, the turret position must be changed by the manual program mode to avoid such obstacles.

As shown in the above figure, the tool path starts from the point A' which is displaced from the point A using the manual program mode.