Interactive Graphic MDI Function

IGF-L3

OSP500L-G/OSP5000L-G CNC SYSTEMS

OPERATION MANUAL

(4th Edition)

OKUMA
Your single source for machine & control

Publication No. CB002
(April 1987)

TABLE OF CONTENTS

Note: To avoid any confusion over the use of the letter "O (oh)" and figure "0 (zero)" in this manual, the numerical value "0 (zero)" is expressed as "Ø" if there is any possibility of misunderstanding.

Interactive Graphic MDI Function

IGF-L3

OSP500L-G/OSP5000L-G CNC SYSTEMS

OPERATION MANUAL — BASIC —

(4th Edition)

OKUMA

Your single source for machine & control

Publication No. CB002 2450-E-R3
(April 1987)

TABLE OF CONTENTS

PAGE

Note 1: To avoid any confusion over the use of the letter "O (oh)" and figure "0 (zero)" in this manual, the numerical value "0 (zero)" is expressed as "Ø" if there is any possibility of misunderstanding.

Note 2: Display on the graphic CRT differs between the color graphic specification and the mono-chrome graphic specification. However, operation procedures are identical for these specifications.

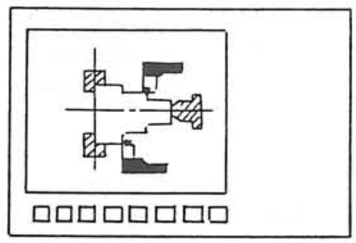
SECTION 1 GENERAL

The Interactive Graphic MDI Function (IGF-L3) has been developed as an extension function of the OSP5ØØØL-G. This function allows easy programming at the OSP5ØØØL-G operation panel through an interactive operation with the NC. By keying-in responses to the prompts displayed on the CRT, the operator can program both parts with complicated contours as well as simple contours with no problems.

Although graphic CRT display differs slightly between the color graphic specifications and the monochrome graphic specifications, operation procedures are identical in both cases.

The IGF-L3 features:

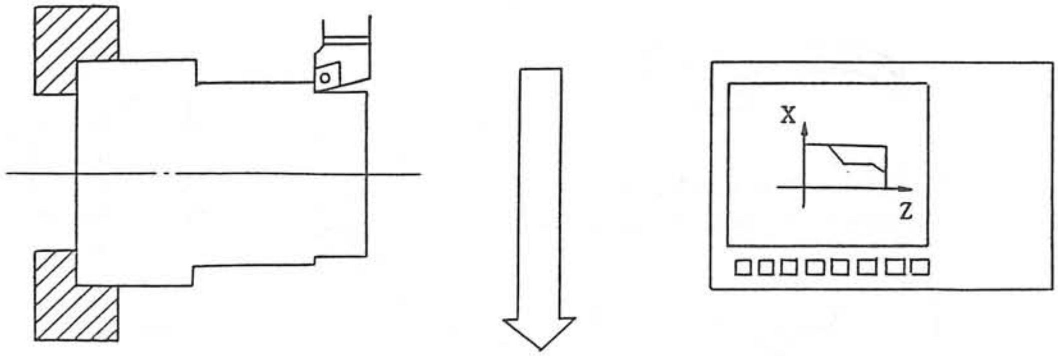
(1) Program check through the animated display (multi-task function)



(2) Automatic programming for simultaneous 4-axis cuts and easy editing of simultaneous 4-axis cut programs

(3) Automatic determination of machining conditions and automatic selection of tools

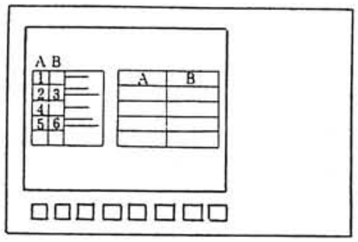
(4) Background programming* (multi-task)



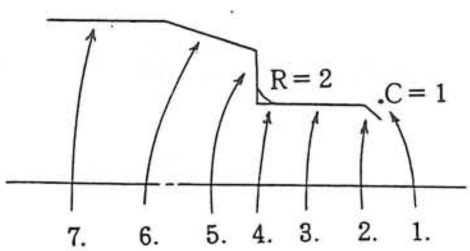
[Actual Machining] Simultaneous Processing [Programming]

* Programming a part is possible while the NC computer is controlling machining processes for other workpiece.

(5) Easy editing of machining order and machining conditions



(6) Easy shape definition



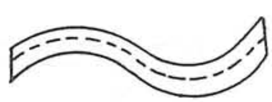
1.	START PT	X=	, Z=
2.	CHF SIZE C	C=	
3.	OD ←	Z=	
4.	RADIUS R	R=	
5.	FACE ↑	X=	
6.	TAPER	X=	, Z=
7.	OD ←	Z=	

(7) Large capacity bubble memory to save input data

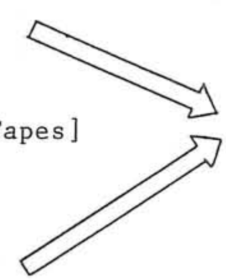
Standard: Capacity equivalent to 60 meter (197 feet) tape length

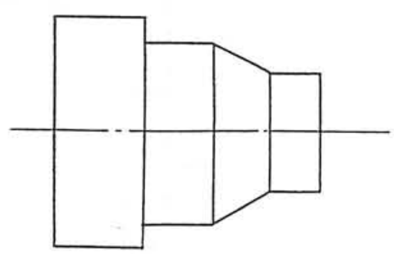
Optional: OSP5000L-G ... Capacity equivalent to 10,240 meter (33,600 feet) tape length
OSP500L-G .... Capacity equivalent to 3,840 meter (12,600 feet) tape length

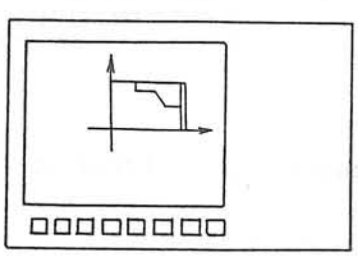
(8) Operations using programs stored from tapes is also possible as well as programs made and stored using the IGF-L3 function.



[Program Storage Using the Conventional Program Tapes]







[IGF-L3 Function]

[Product]

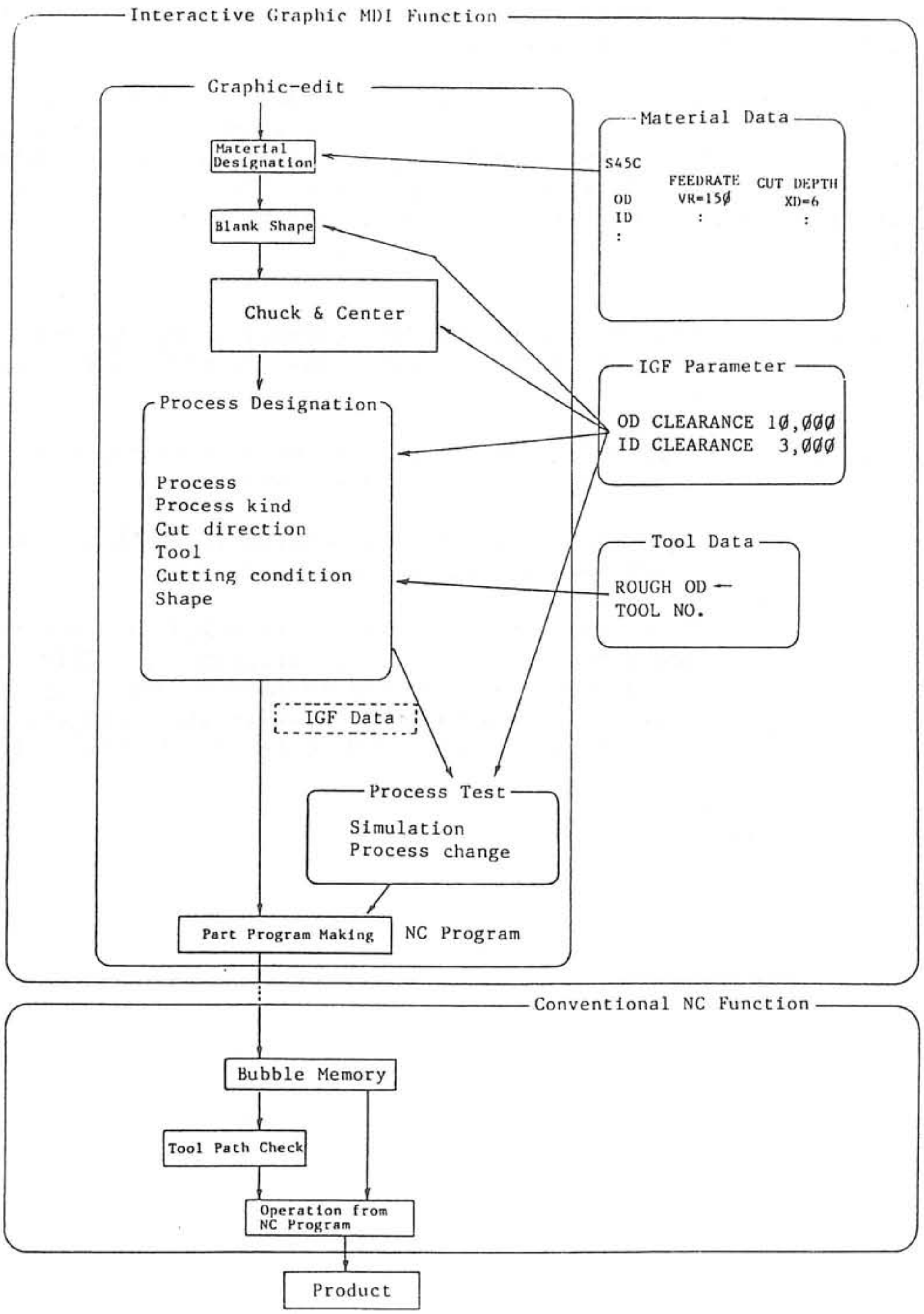
SECTION 2 OUTLINE OF FUNCTIONS

The IGF-L3 executes the machining test (graphic display of the machining status) when the IGF data (shape, cutting conditions, tools, etc. required for programming a part) is entered using the graphic editing function. The data of the machining order and cutting conditions is converted into NC programs after verification and editing.

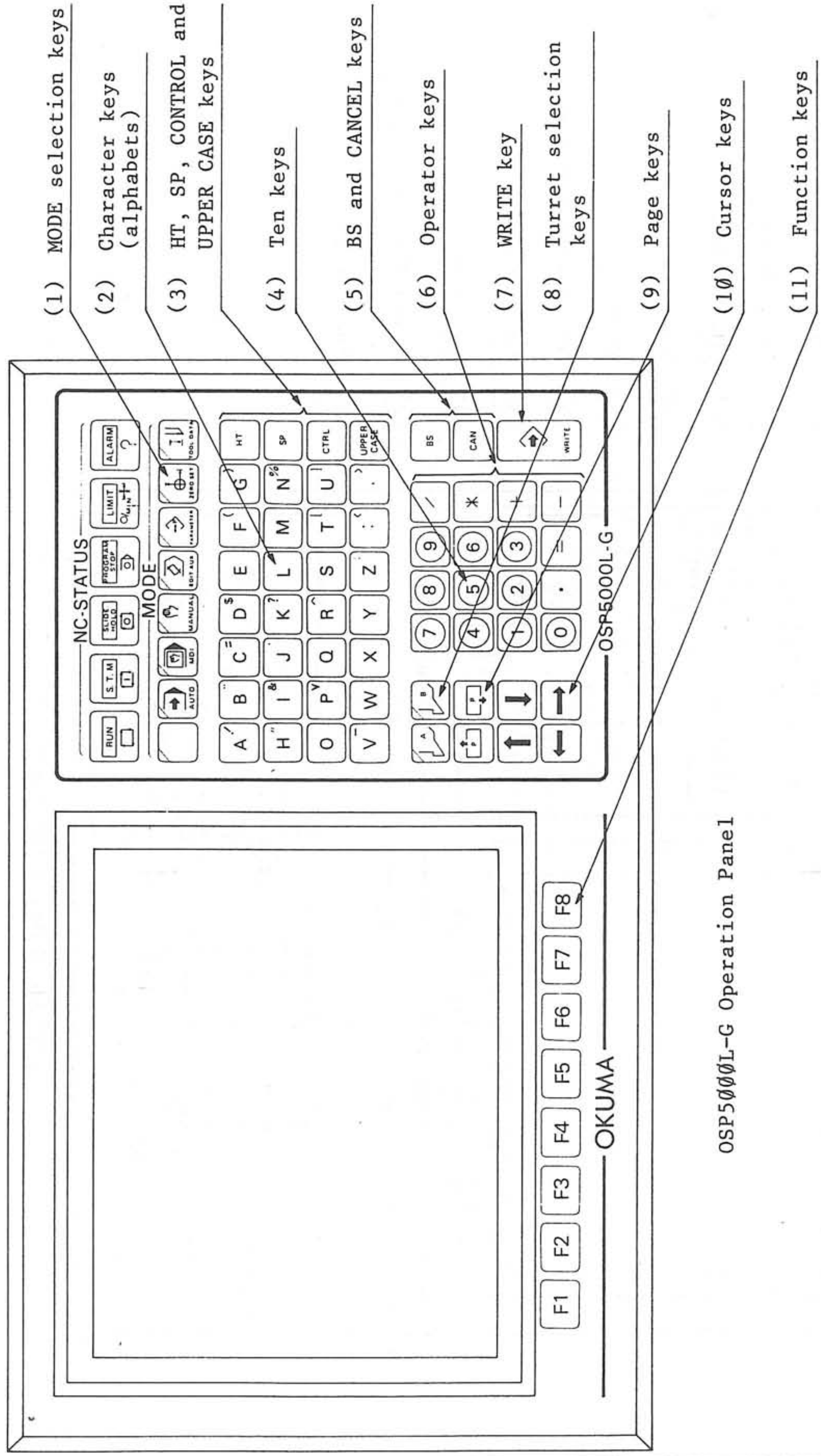
Tool data, material data and parameters are automatically called when the entry of them is required during programming a part by entering the data in response to the guides (guide drawings and guide messages) displayed on the CRT. Then, they are taken into the program. The data indicated above (tool data, material data and parameters) can be separately registered and edited.

IGF data :	This refers to the data entered using the graphic editing function such as blank shape, cutting conditions, tools, work shape, etc.
Tool data :	This refers to the data related to tools, such as tool type, tool shape and turret number.
Material data :	This refers to the data related to the blank, such as blank material and cutting conditions.
IGF parameters:	This refers to the data not included in the classification above but requisite for NC programming. This includes measurements of standard tailstock center, upper and lower limit rpms in individual spindle speed ranges, and designation of nominal diameter for thread cutting.

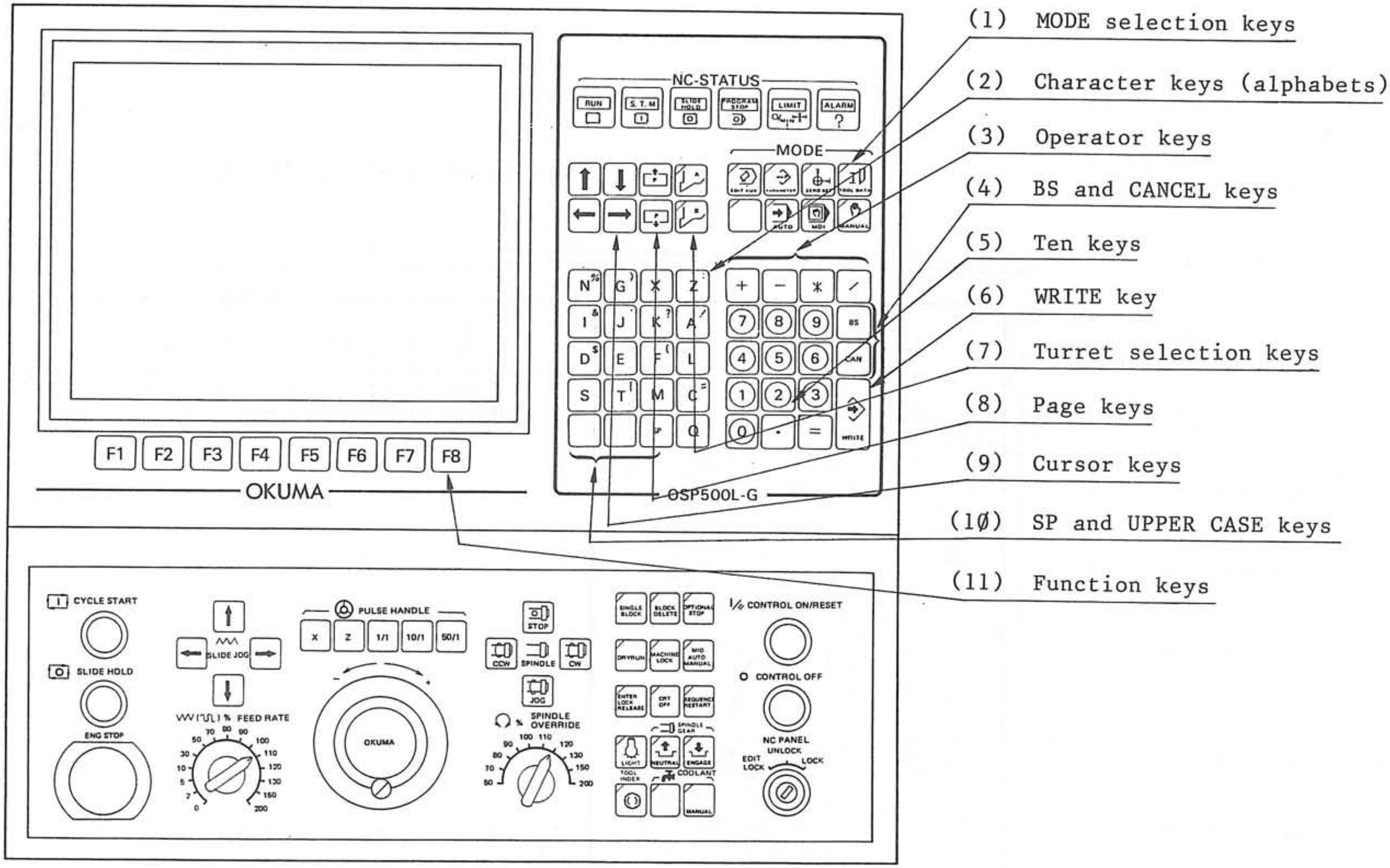
General Flow of IGF-L3 Programming:



SECTION 3 OPERATION PANEL AND KEYBOARD



OSP5ØØØL-G Operation Panel



OSP5ØØL-G Operation Panel

Explanations are given assuming the OSP5ØØØL-G operation panel. Operations with OSP5ØØL-G are identical to those with OSP5ØØØL-G although arrangement of keys and switches on the operation panel differs each other.

SECTION 4 DATA SETTING PROCEDURE

Tool data, material data and IGF parameters are factory-set at Okuma for general turning operations. However, they can be edited as needed so that user's knowhow can be reflected in programming.

4-1. TOOL DATA

Tools to be used can be registered before beginning programming. The IGF system can register the tool data of up to 26 tools and data of standard 12 tools is provided with the system.

The tool data consists of following three parameters:

DATA NO., TOOL CODE, and TOOL FORM CODE

(1) DATA NO. (NO)

A total of 26 tools can be registered and the data number is used for identifying the individual tools.

For two-saddle models, a total of 26 tools can be registered for individual turrets.

(2) TOOL CODE (CD)

The tools are classified in accordance with the type and direction of cutting. The system has 26 combinations of these parameters and one code number is assigned to one tool number.

(3) TOOL FORM CODE (FC)

The IGF system displays several possible tools in accordance with the entered TOOL CODE. The operator has an option to select the most desirable tool to use for intended cutting depending on the mounting direction of tool and tool shape. One tool number is assigned with one tool form code number.

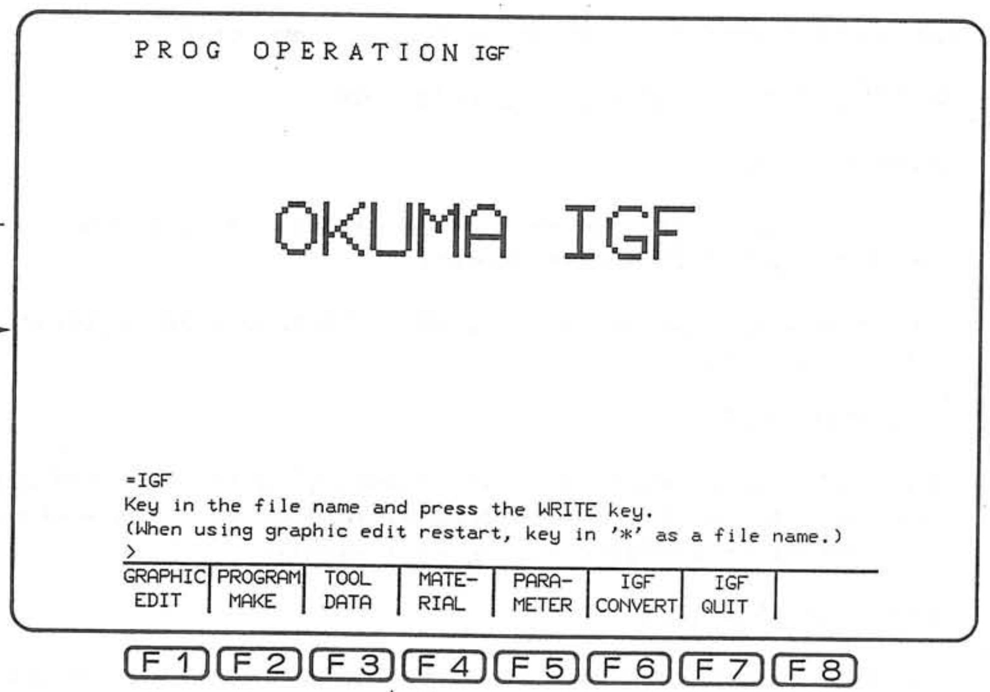
IGF Tool Data Entry Processes

1) From the program operation mode, press the function key [F7] (IGF).

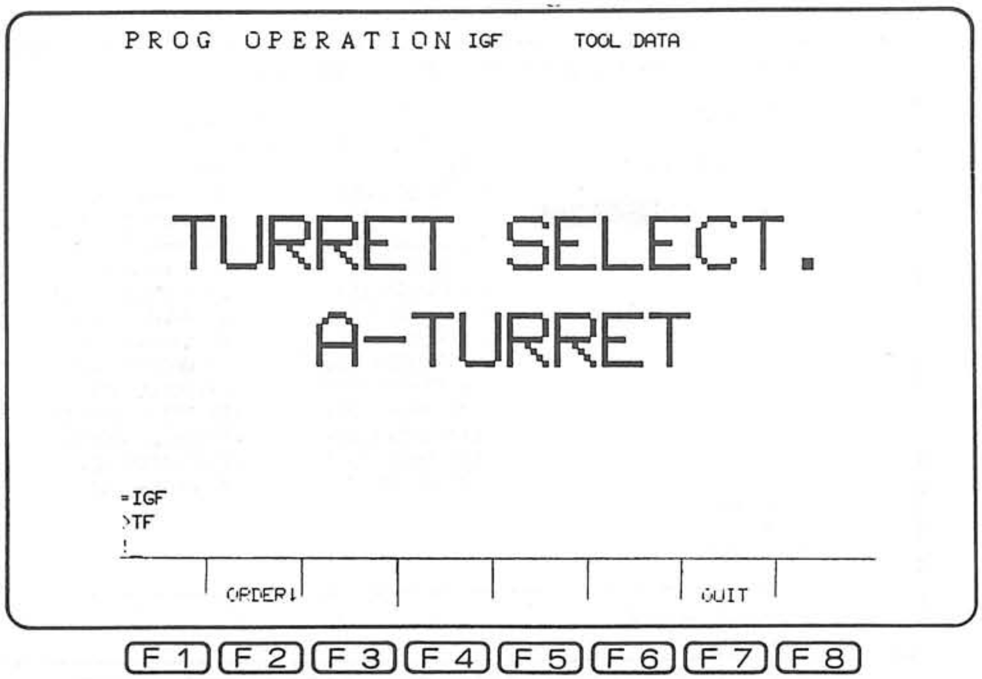
[IGF]

2) After confirming the display of the IGF start-up page (OKUMA IGF), press the function key [F3] (TOOL DATA).

[F7]
(QUIT)



3) Select the turret (for 2S models).

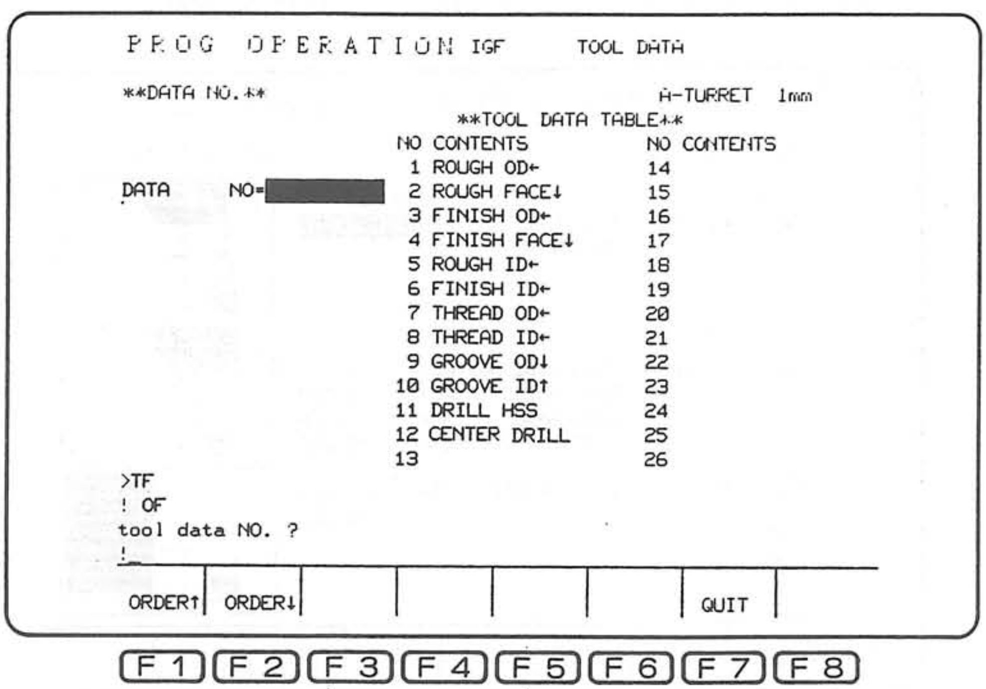


[F7]
(QUIT)

[F1] (ORDER ↑)

[F2] (ORDER ↓)

4) Select the TOOL DATA NO. (1 - 26).



[F7]
(QUIT)

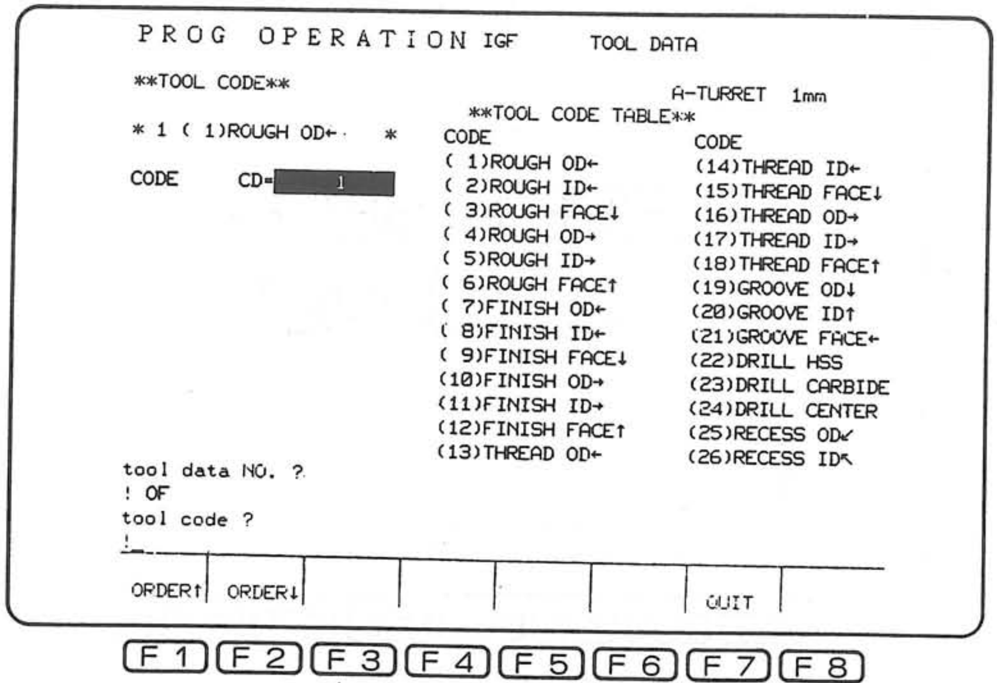
[F1] (ORDER ↑)

[F2] (ORDER ↓)

[F1] (ORDER ↑) [F2] (ORDER ↓)

5) Select the TOOL CODE NO. (CD) (1 - 26).

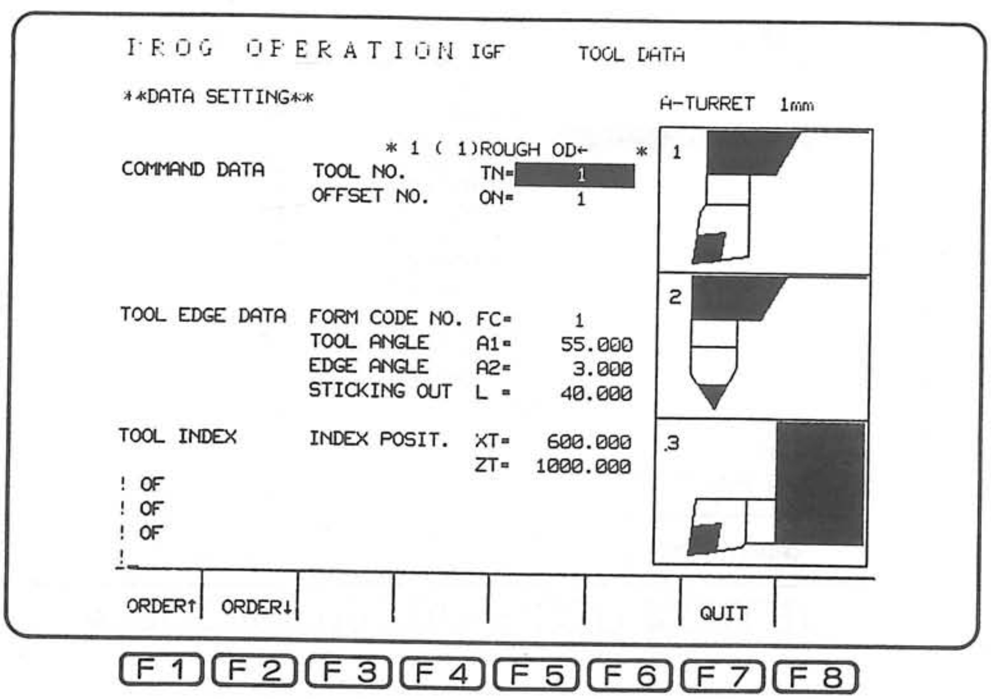
[F7] (QUIT)



[F1] (ORDER ↑) [F2] (ORDER ↓)

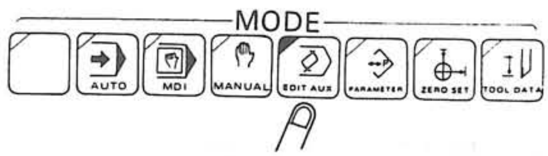
6) Enter the data for parameters displayed on the CRT. Enter the FORM CODE NO. (FC) in reference to the guide drawing.

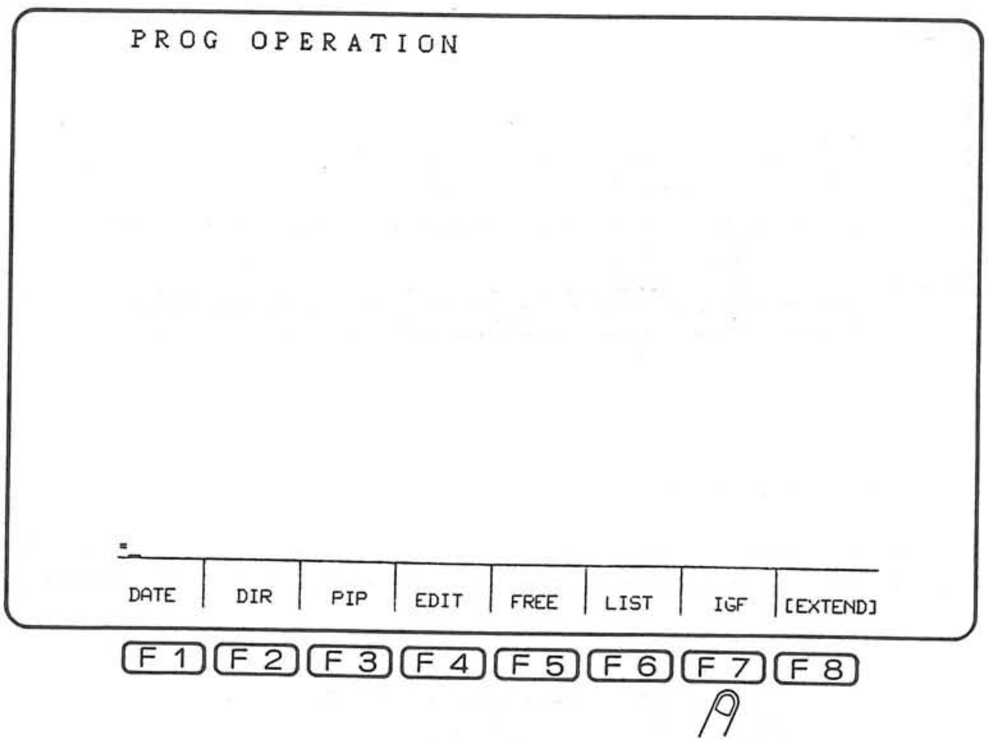
[F7] (QUIT)



Example of Tool Data Entry

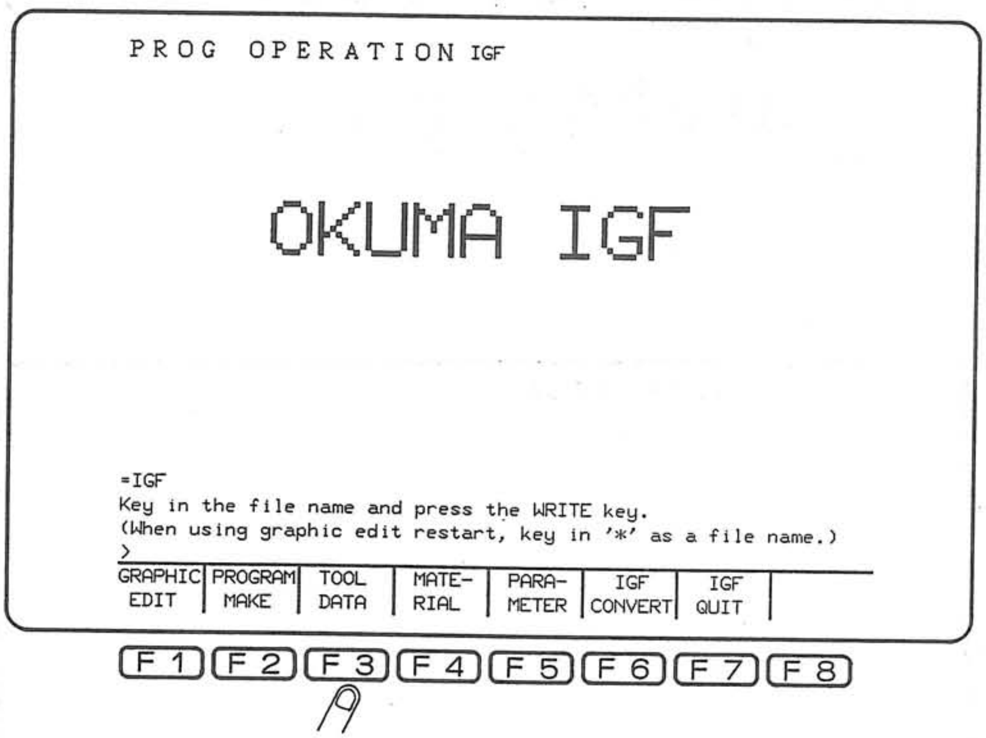
1) Select the PROGRAM OPERATION mode by pressing the EDIT AUX. key.





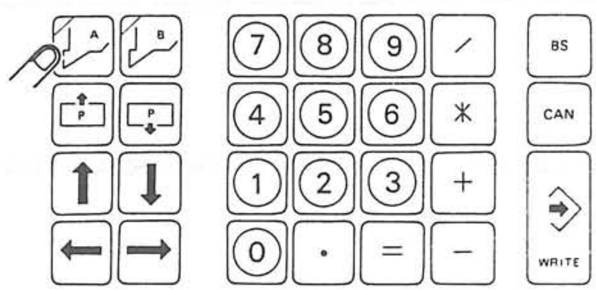
2) Press the function key [F7] (IGF) to select the IGF mode.

The IGF start-up page will appear.

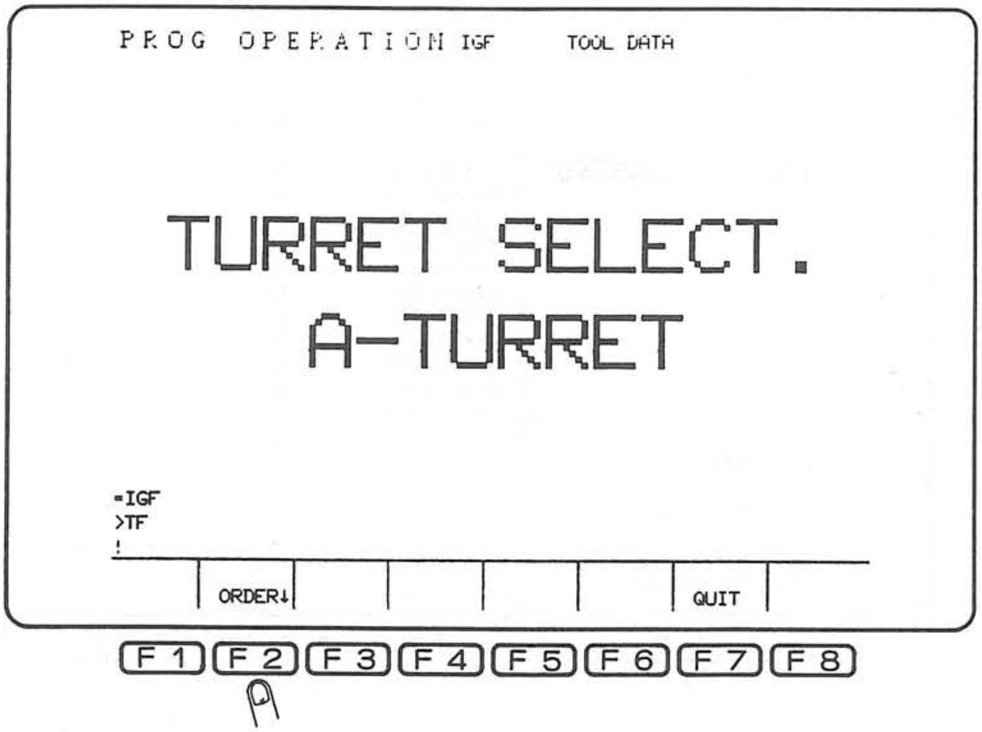


3) Press the function key [F3] (TOOL DATA).

4) For the 2S model, select the turret (A or B). Turret selection is possible by pressing the proper turret keys on the NC operation panel.

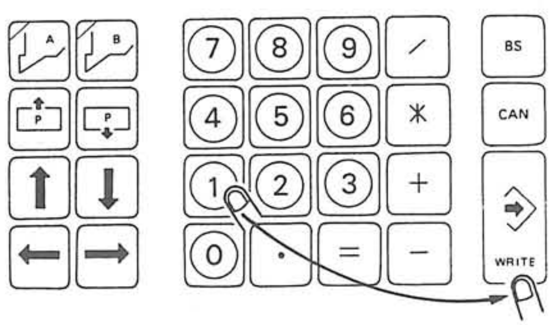


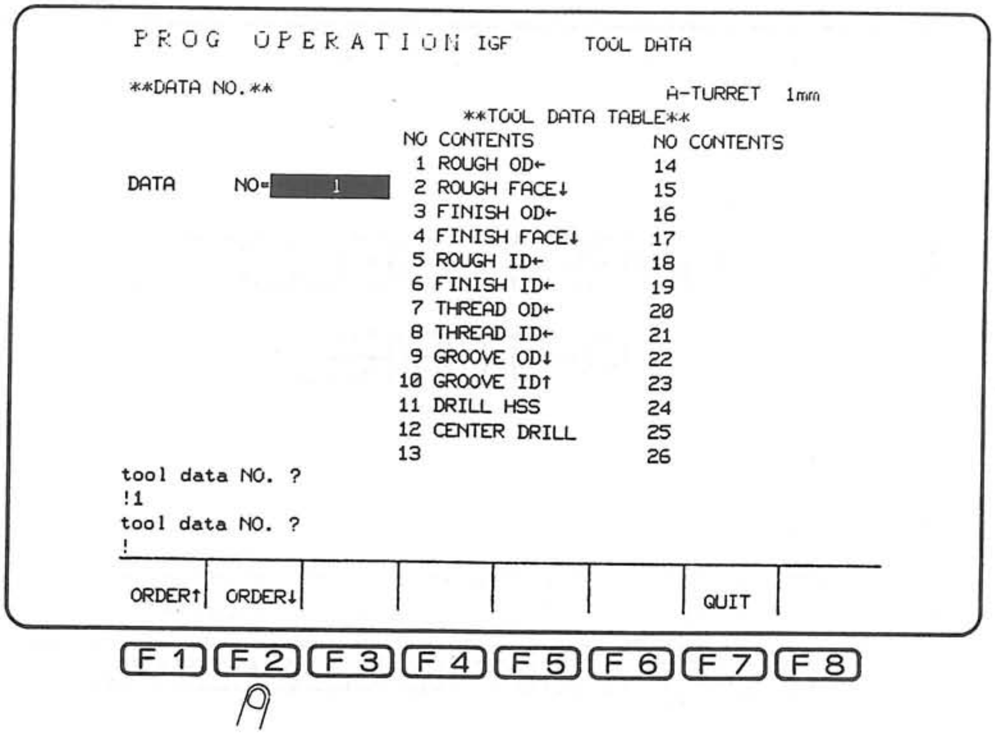
5) The CRT will display the turret selected.



After confirming the display of the selected turret, press the function key [F2] (ORDER ↓).

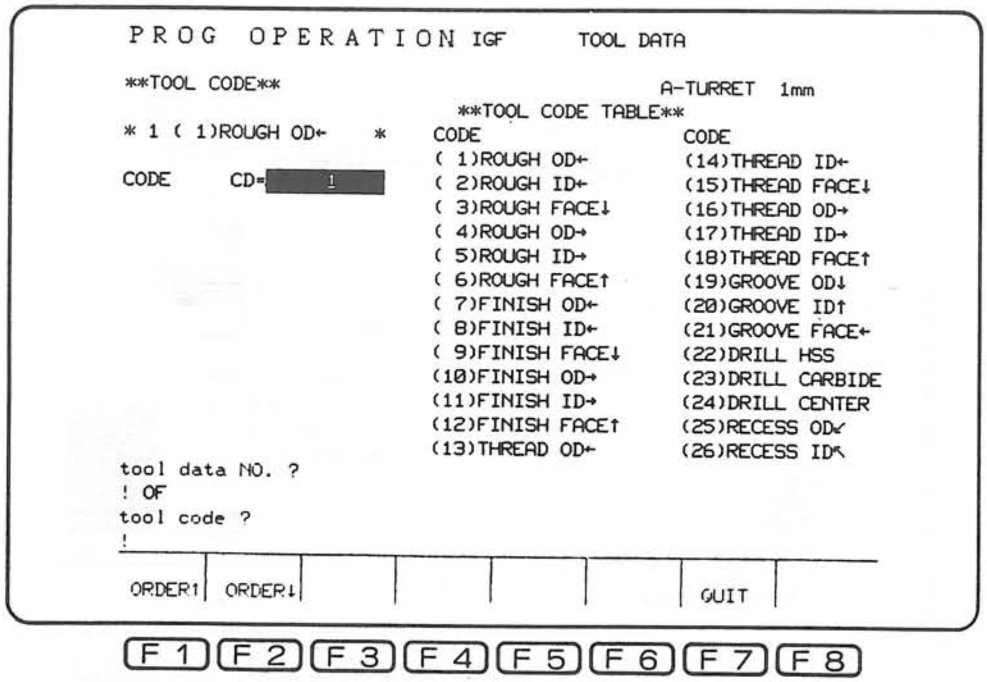
6) The page is changed to the **DATA NO.** page of the TOOL DATA. Select the ROUGH OD ← tool by entering "1".



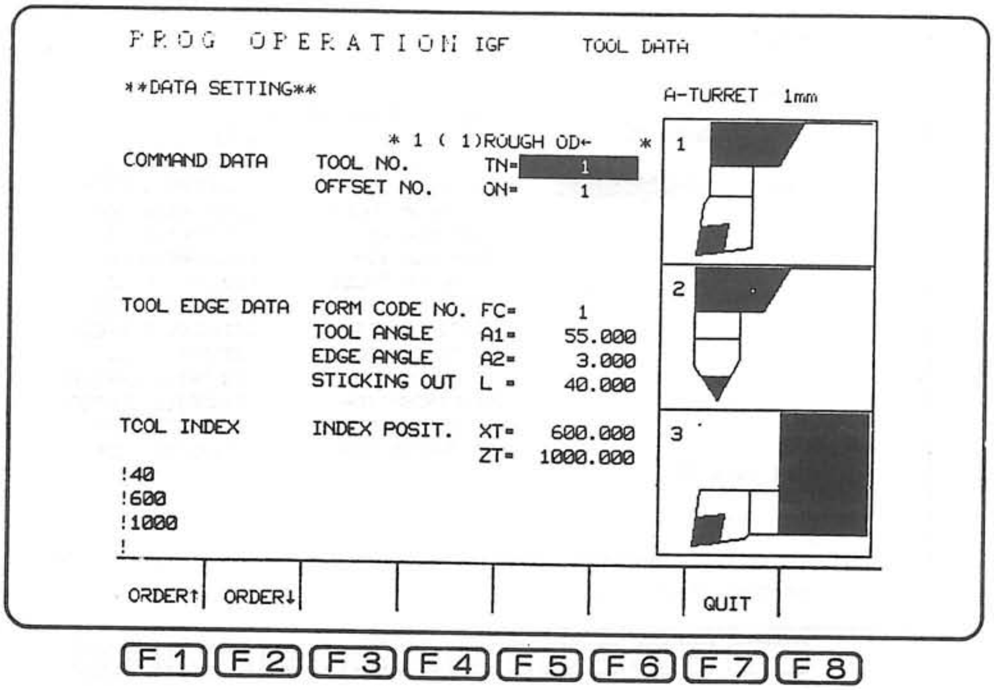


7) After making sure the display of the entered tool number "1", press the function key [F2] (ORDER ↓).

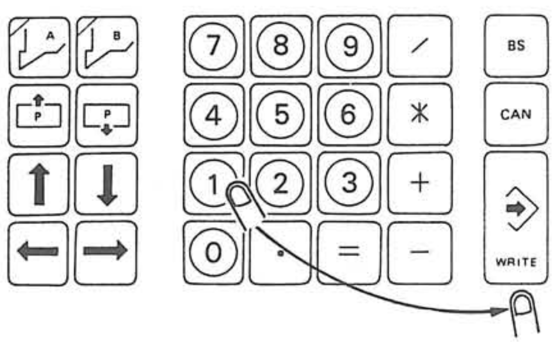
8) The page is changed to the **TOOL CODE** page of the TOOL DATA. Select the ROUGH OD ← by entering "1". After making sure the desired tool code has been selected, press the function key [F2] (ORDER ↓).



9) The page is changed to the **DATA SETTING** page of the TOOL DATA. Enter the data for parameters displayed including TOOL NO., OFFSET NO. and FORM CODE NO.



Enter "1" for parameter TOOL NO.



Enter the data for remaining parameters in the similar manner. For the entry of FORM CODE NO., select the data to be entered from the guide drawing displayed.

4-2. MATERIAL DATA

Cutting conditions are registered for individual blank materials. A total of 13 types of material data can be registered and the IGF system is delivered with eight standard type material data.

The material data consists of the following three parameters:

NAME NO., MATERIAL NAME, and CUTTING CONDITIONS

(1) NAME NO.

A total of 13 different materials can be registered. The NAME NO. sets the number for individual materials.

(2) MATERIAL NAME

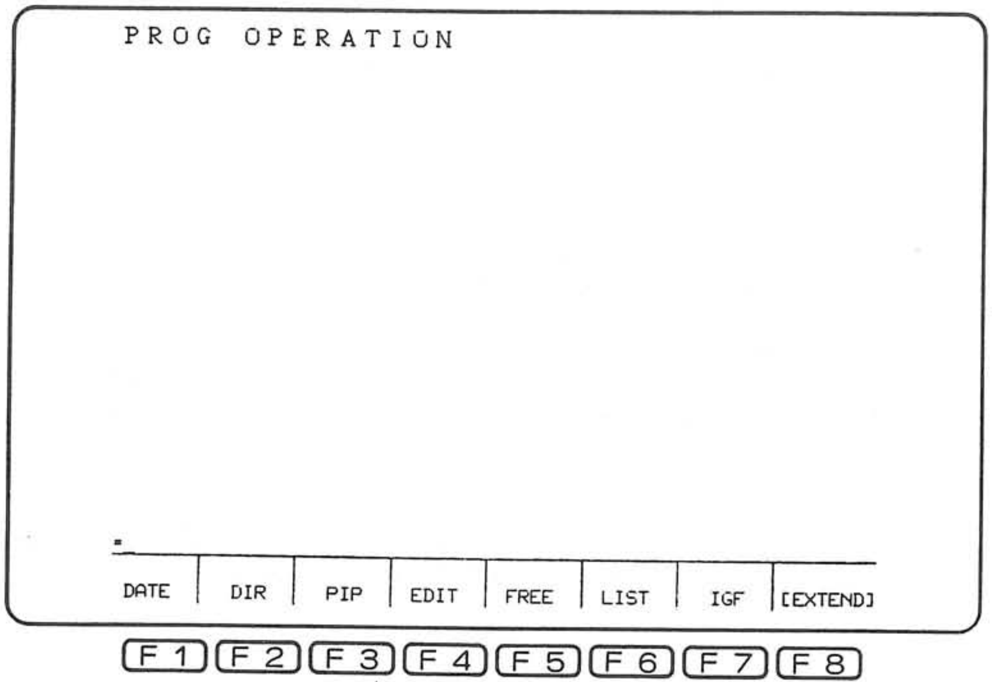
Material name can be registered in up to eight alphabets or numbers.

(3) CUTTING CONDITIONS

Cutting conditions for individual tools are registered. The parameters to be set include CUT. SPEED and FEEDRATE.

IGF Material Data Entry Processes

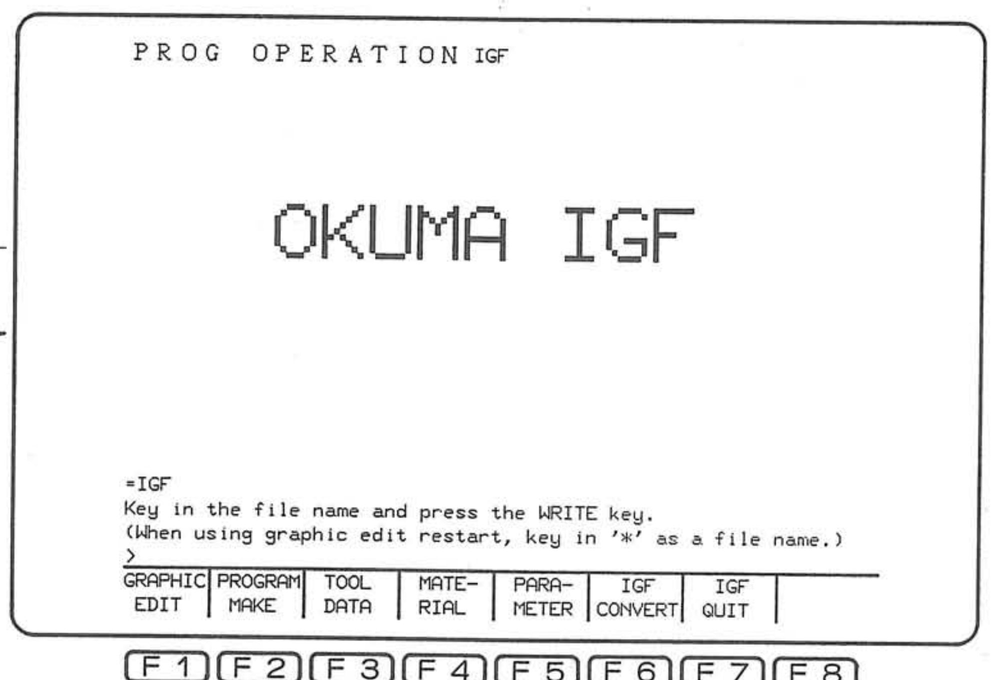
1) From the program operation mode, press the function key [F7] (IGF).



[F7] (IGF)

2) After confirming the display of the IGF start-up page (OKUMA IGF), press the function key [F4] (MATERIAL).

[F7]
(QUIT)



[F4] (MATERIAL)

[F4] (MATERIAL)

3) Select the material number in reference to the **MATERIAL TABLE** and enter the corresponding number.

```
PROG OPERATION IGF      MAT. DATA

 - 1-
                              **MATERIAL TABLE**
                          NO NAME
                            1 S10C
NAME      NO=               2 S45C
                            3 SCM4
                            4 SUS304
                            5 FC20
                            6 FCD45
                            7 AL
                            8 BC3
                            9
                           10
                           11
                           12
                           13
=IGF
>MF
material name NO. ?
!_
ORDER↑ | ORDER↓ |  |  |  |  | QUIT |
```

F 1 | F 2 | F 3 | F 4 | F 5 | F 6 | F 7 | F 8

[F7] (QUIT)

[F1] (ORDER ↑)

[F2] (ORDER ↓)

4) Enter material name.

```
PROG OPERATION IGF      MAT. DATA

 - 2-
  NO 2=S45C                *MATERIAL NAME*

NAME      NA=

                           material name within 8 alphanumerical

                           characters can be keyed in.

!2
material name NO. ?
! OF
!_
ORDER↑ | ORDER↓ |  |  |  |  | QUIT |
```

F 1 | F 2 | F 3 | F 4 | F 5 | F 6 | F 7 | F 8

[F7] (QUIT)

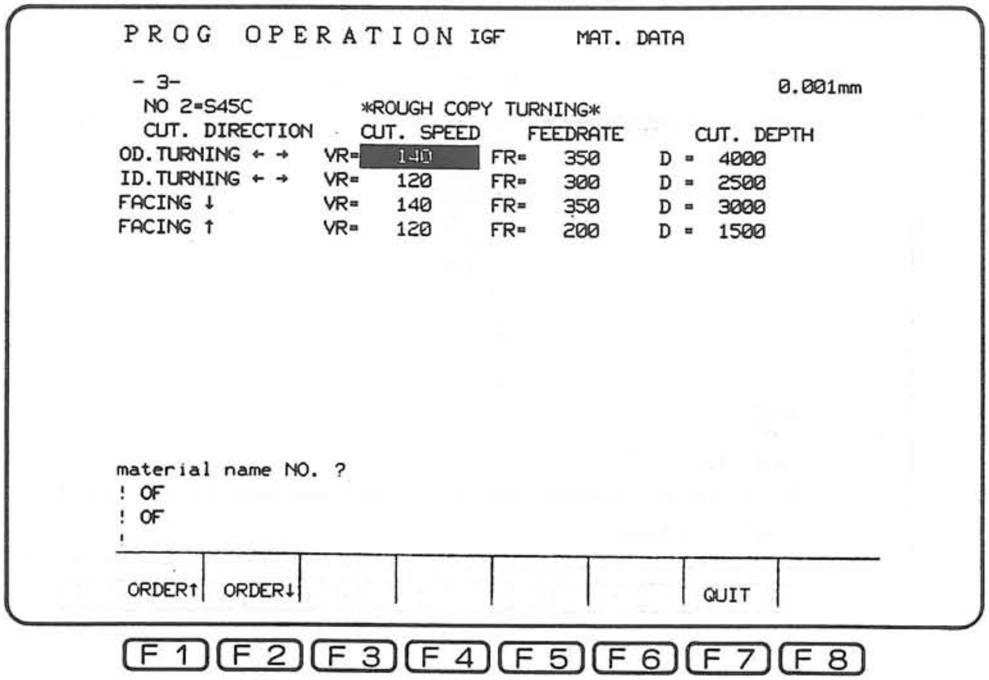
[F1] (ORDER ↑)

[F2] (ORDER ↓)

[F1] (ORDER ↑) [F2] (ORDER ↓)

5) The cutting condition list is displayed. Enter the data as required.

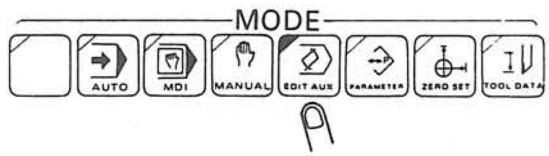
[F7] (QUIT)

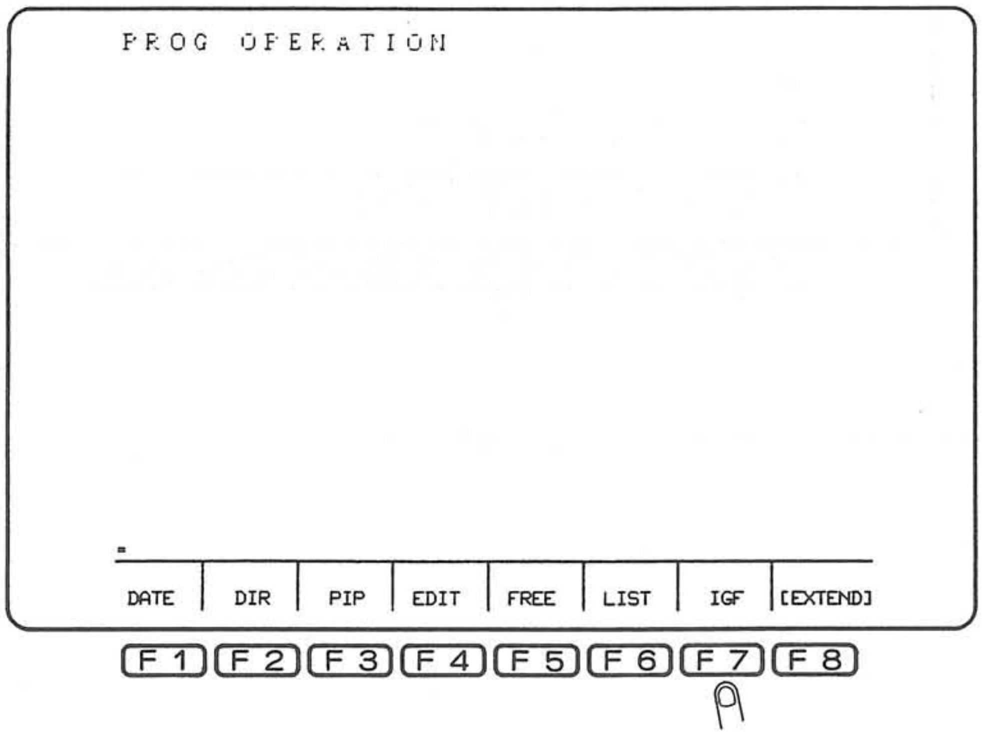


Example of Material Data Entry

The procedure to enter NAME NO=3 and NAME NA=SCM4 is explained below:

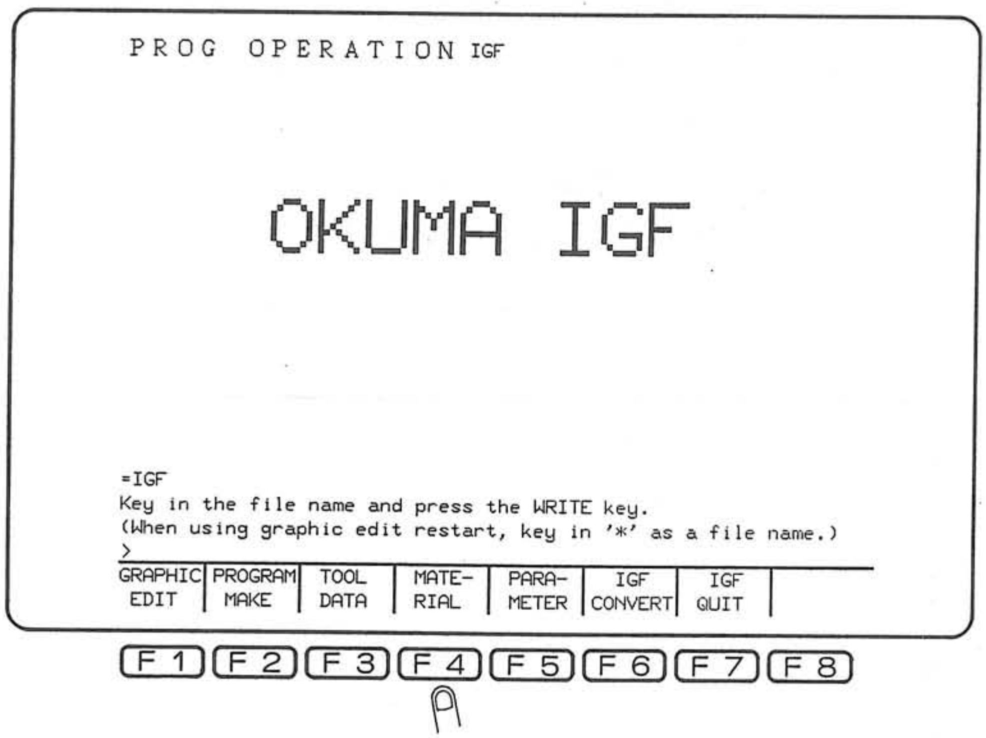
1) Select the PROGRAM OPERATION mode by pressing the EDIT AUX. key.





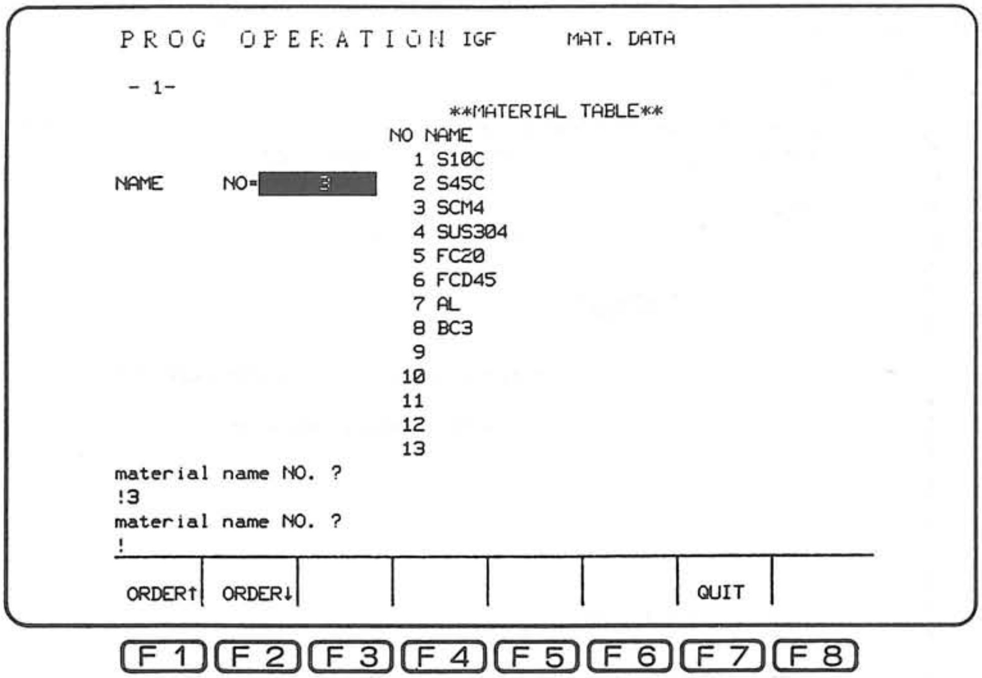
2) Press the function key [F7] (IGF) to select the IGF mode.

The IGF start-up page will appear.

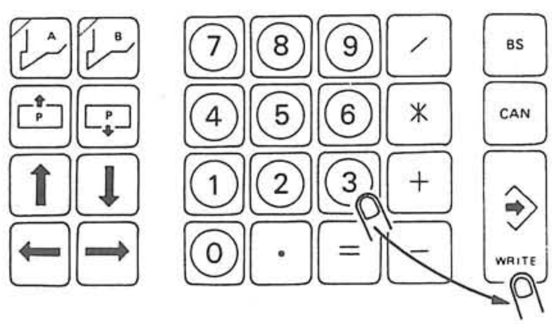


3) Press the function key [F4] (MATERIAL).

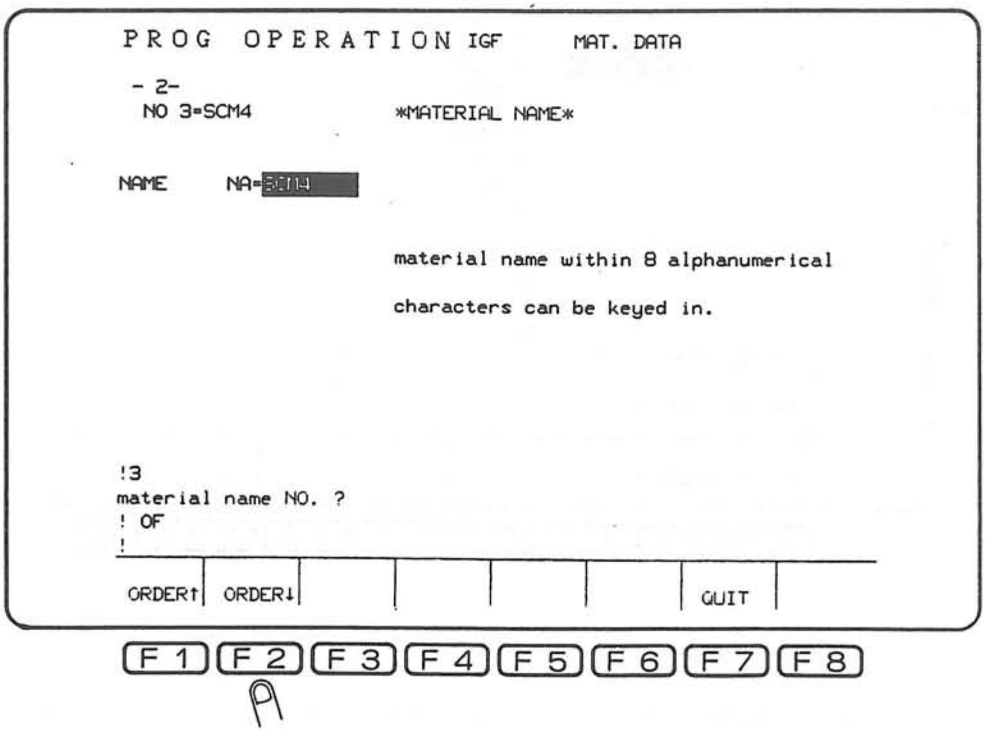
4) The page is changed as indicated below.



Select the number from the **MATERIAL TABLE**. In this example, select "3" for SCM4. Key in the material number through the operation panel.



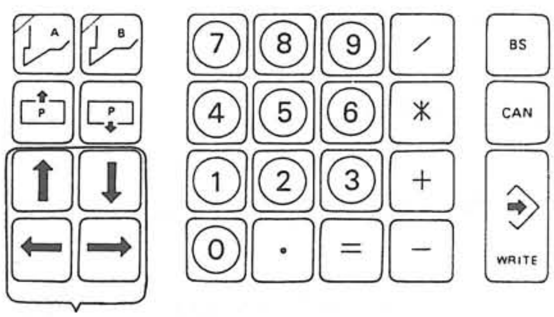
5) The page is changed to the *MATEIRAL NAME* page. The operator can enter the material name as required. In this example, the IGF system will display the default "SCM4". Since the entry of material name is not necessary, press the function key [F2] (ORDER ↓).



6) The page is changed to the cutting data entry page.

```
PROG  OPERATION IGF     MAT. DATA

 - 3-                                                0.001mm
  NO 3=SCM4          *ROUGH COPY TURNING*
  CUT. DIRECTION     CUT. SPEED    FEEDRATE       CUT. DEPTH
OD.TURNING ← →     VR=  100      FR=  350      D =  4000
ID.TURNING ← →     VR=   80      FR=  300      D =  2500
FACING ↓           VR=  100      FR=  350      D =  3000
FACING ↑           VR=   80      FR=  200      D =  1500




material name NO. ?
! OF
! OF
!
ORDER↑ | ORDER↓ |  |  |  |  | QUIT |
 F 1     F 2    F 3  F 4  F 5  F 6  F 7   F 8
```

For entering the data, locate the cursor to the desired parameter data position using the cursor keys. Then, enter the data from the keyboard.



Cursor keys

In this example, since the IGF system displays the default which has been registered as the standard data for SCM4, no entry is made from this page.

7) Press the function key [F7] (QUIT).

```
!
ORDER↑ | ORDER↓ |  |  |  |  | QUIT |
 F 1     F 2    F 3  F 4  F 5  F 6  F 7   F 8
```

Material Data Table

The data for individual materials is registered in accordance with the units indicated in the table below. The data size is from "Ø" to "32ØØ" and entry of a decimal point is not allowed. Data entry should be made taking this into consideration.

Materials and cutting conditions supplied with the IGF system are the standard conditions and they cannot be always used for any type of turning operations. Depending on cutting shape, work shape, chucking setup and other factors, the data will be required to be edited meeting the specific turning requirements.

Material Data Table

Cutting Process	Cutting Conditions		Unit	
			Metric	Inch
ROUGH COPY TURNING	CUT. SPEED	VR	m/min	feet/min
	FEEDRATE	FR	μm/rev	1/1ØØØØ inch/rev
	CUT. DEPTH	DX (Z)	μm	1/1ØØØØ inch/rev
FINISHING	CUTTING SPEED	VF	m/min	feet/min
	FINISHING STOCK X-AXIS	LX	μm	1/1ØØØØ inch
	FINISHING STOCK Z-AXIS	LZ	μm	1/1ØØØØ inch
	FINISH ▽ (ROUGH) FEEDRATE	F1	μm/rev	1/1ØØØØ inch/rev
	FINISH ▽▽ (SEMI FINISH) FEEDRATE	F2	μm/rev	1/1ØØØØ inch/rev
	FINISH ▽▽▽ (FINISH) FEEDRATE	F3	μm/rev	1/1ØØØØ inch/rev
	FINISH ▽▽▽▽ (FINE FINISH) FEEDRATE	F4	μm/rev	1/1ØØØØ inch/rev
THREAD CUTTING	CUT. SPEED	V	m/min	feet/min
	CUT. DEPTH	D	μm	1/1ØØØØ inch
	FIN. STOCK	L	μm	1/1ØØØØ inch
THREAD TAPPING	CUT. SPEED	V	m/min	feet/min

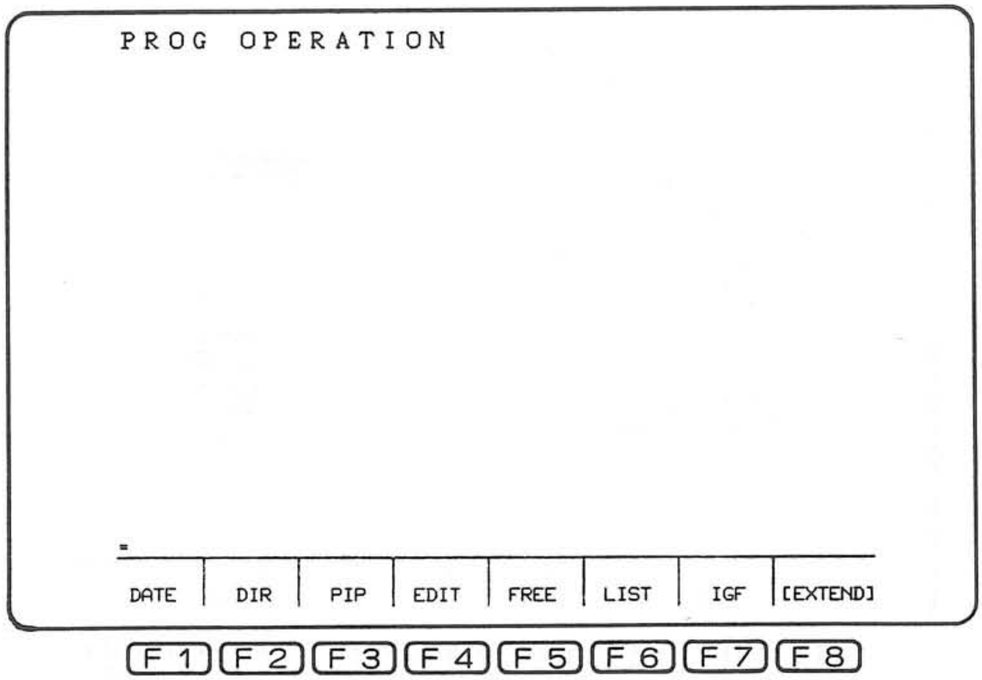
Cutting Process	Cutting Conditions		Unit	
			Metric	Inch
GROOVING	CUT. SPEED	VR	m/min	feet/min
	FEEDRATE	FR	μm/rev	1/1ØØØØ inch/rev
	CUT. DEPTH	D	μm	1/1ØØØØ inch/rev
	CUTTING SPEED	VF	m/min	feet/min
	FINISHING STOCK X-AXIS	LX	μm	1/1ØØØØ inch
	FINISHING STOCK Z-AXIS	LZ	μm	1/1ØØØØ/ inch
	FINISH ∇ (ROUGH) FEEDRATE	F1	μm/rev	1/1ØØØØ inch/rev
	FINISH ∇∇ (SEMI FINISH) FEEDRATE	F2	μm/rev	1/1ØØØØ inch/rev
	FINISH ∇∇∇ (FINISH) FEEDRATE	F3	μm/rev	1/1ØØØØ inch/rev
	FINISH ∇∇∇∇ (FINE FINISH) FEEDRATE	F4	μm/rev	1/1ØØØØ inch/rev
DRILLING (HSS)	CUT. SPEED	V	m/min	feet/min
	FEEDRATE	F	μm/rev	1/1ØØØØ inch/rev
	CUT. DEPTH	D	μm	1/1ØØØØ inch/rev
DRILLING (CARBIDE)	CUT. SPEED	V	m/min	feet/min
	FEEDRATE	F	μm/rev	1/1ØØØØ inch/rev
	CUT. DEPTH	D	μm	1/1ØØØØ inch/rev
DRILLING	SPINDLE RPM	S	rpm	rpm
	FEEDRATE	F	μm/rev	1/1ØØØØ inch/rev
	DEPTH OF CUT	D	μm	1/1ØØØØ inch
CENTERING	SPINDLE RPM	S	rpm	rpm
	FEEDRATE	F	μm/rev	1/1ØØØØ inch/rev
RECESSING	SPINDLE SPEED	V	m/min	feet/min
	FEEDRATE	F	μm/rev	1/1ØØØØ inch/rev

4-3. IGF PARAMETERS

The IGF system has various parameters including dimension parameters, integer parameters and percent parameters.

IGF Parameter Data Entry Processes

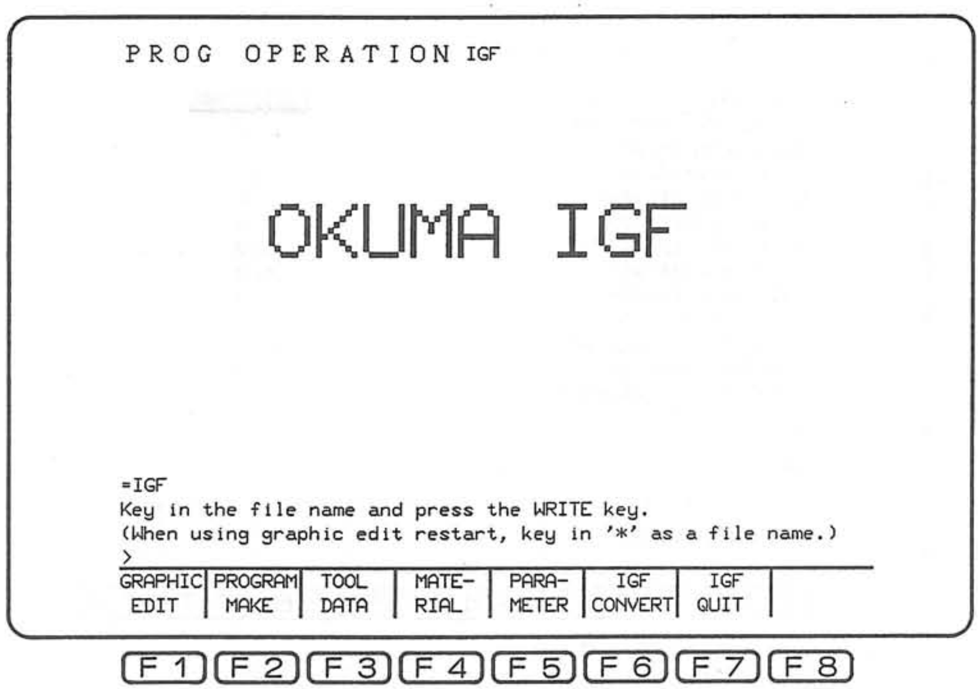
1) From the program operation mode, press the function key [F7] (IGF).



[F7] (IGF)

2) After confirming the display of the IGF start-up page (OKUMA IGF), press the function key [F5] (PARAMETER).

[F7]
(QUIT)



[F5] (PARAMETER)

[F5] (PARAMETER)

3) Set the data for dimension parameters.

[F7] (QUIT)

```
PROG OPERATION IGF      PARAMETER
 - 1 -                                              1mm
              *DIMENSION PARAMETER*
 1 OD CLEARANCE                          10.000
 2 ID CLEARANCE                           3.000
 3 FACE CLEARANCE                         3.000
 4 BLANK INSIDE LENGTH IL                 0.000
 5 BLANK INSIDE DIA. ID                   0.000
 6 DIMENSION ZERO SHIFT                   0.000
 7 PROGRAM ZERO SHIFT                     0.000
 8 NOSE-R CANCEL TRAVEL                   0.010
 9 CENTER LENGTH L2                      10.000
10 CENTER DIA. D2                        36.000
11 CENTER HOLE DIA. D3                    5.000
12 GROOVING CLEARANCE                     0.100
13
>PF
! OF
! OB
!_
       | ORDER↓ |      |      |      |      | QUIT |
```

F1 F2 F3 F4 F5 F6 F7 F8

[F1] (ORDER ↑) [F2] (ORDER ↓)

4) Set the data for integer parameters.

[F7] (QUIT)

```
PROG OPERATION IGF      PARAMETER
 - 2-
              *INTEGER PARAMETER*
 1 SEQUENCE NO. INCREMENT                   10
 2 MATERIAL CODE DIGITS                      2
 3 M41 MINIMUM RPM                          65
 4 M42 MINIMUM RPM                         205
 5 M43 MINIMUM RPM                           0
 6 M44 MINIMUM RPM                           0
 7 M41 MAXIMUM RPM                        1110
 8 M42 MAXIMUM RPM                        3500
 9 M43 MAXIMUM RPM                           0
10 M44 MAXIMUM RPM                           0
11 MAXIMUM SPINDLE RPM                    3500
12 ORDER PARAMETER                           5
13 TOOL DATA PARAMETER                       1
=IGF
>PF
! OF
!
ORDER↑ | ORDER↓ |      |      |      |      | QUIT |
```

F1 F2 F3 F4 F5 F6 F7 F8

[F1] (ORDER ↑) [F2] (ORDER ↓)

[F1] (ORDER ↑) [F2] (ORDER ↓)

5) Set the data for percent parameters - spindle control parameters, tool change control parameters, drilling parameters, grooving parameters, coolant control parameters and bar feeder/bar puller parameters.

[F7]
(QUIT)

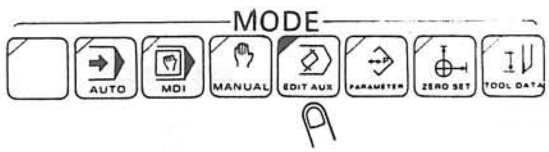
```
PROG OPERATION IGF          PARAMETER

- 5-
                *PERCENT PARAMETER*

 1 ROUGHING FEEDRATE↖                          150
 2 ROUGHING FEEDRATE←                          100
 3 ROUGHING FEEDRATE↙                          100
 4 COPYING FEEDRATE↖                           150
 5 COPYING FEEDRATE←                           100
 6 COPYING FEEDRATE↙                           100
 7 GROOVING SHIFT                               90
 8 GROOVING MID RETRACT                        200
 9 GROOVING END DWELL (RPM)                    200
10 DRILLING MID RETRACT                        200
11 THRU HOLE SLOW FEED CUT DEPTH                50
12 THRU HOLE SLOW FEEDRATE                      50
13 DRILLING END DWELL (RPM)                    200
! OF
! OF
! OF
!
ORDER↑ | ORDER↓ |  |  |  |  | QUIT |
```

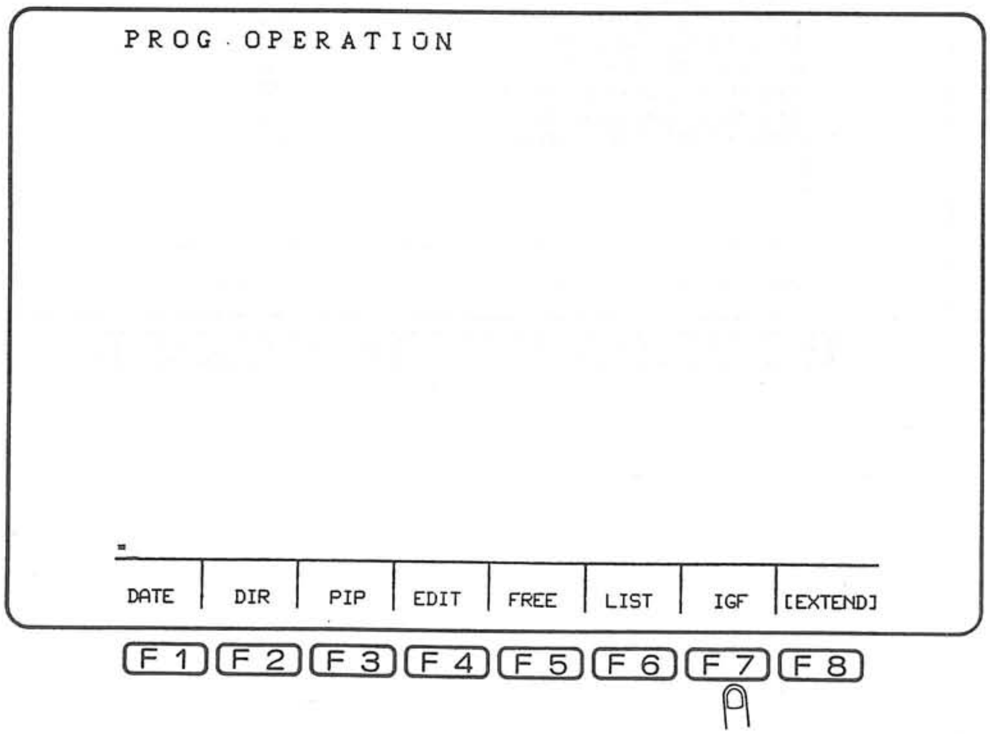
F 1 | F 2 | F 3 | F 4 | F 5 | F 6 | F 7 | F 8

Example of Parameter Data Entry

The procedure to set the sequence number assigning parameter data so that sequence numbers may be assigned in the consecutive order is explained below:

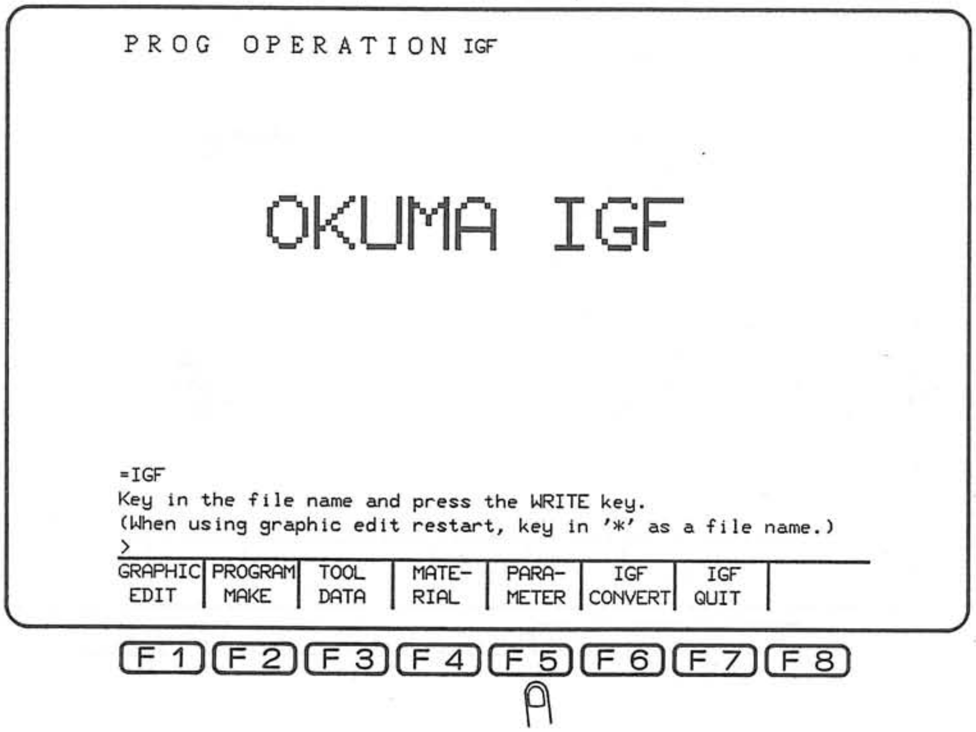
1) Select the PROGRAM OPERATION mode by pressing the EDIT AUX. key.





2) Press the function key [F7] (IGF) to select the IGF mode.

The IGF start-up page will appear.



3) Press the function key [F5] (PARAMETER).

The CRT will display the *DIMENSION PARAMETER* page.

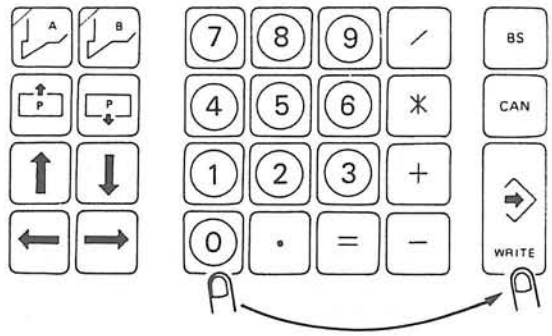
```
PROG  OPERATION IGF      PARAMETER
 - 1 -                                                  1mm
                    *DIMENSION PARAMETER*
 1 OD CLEARANCE                              10.000
 2 ID CLEARANCE                               3.000
 3 FACE CLEARANCE                             3.000
 4 BLANK INSIDE LENGTH IL                     0.000
 5 BLANK INSIDE DIA. ID                       0.000
 6 DIMENSION ZERO SHIFT                       0.000
 7 PROGRAM ZERO SHIFT                         0.000
 8 NOSE-R CANCEL TRAVEL                       0.010
 9 CENTER LENGTH L2                          10.000
10 CENTER DIA. D2                            36.000
11 CENTER HOLE DIA. D3                        5.000
12 GROOVING CLEARANCE                         0.100
13
>PF
! OF
! OB
!_
      | ORDER↓ |     |     |     |     | QUIT |
 F1     F2       F3    F4    F5    F6    F7     F8
```

A

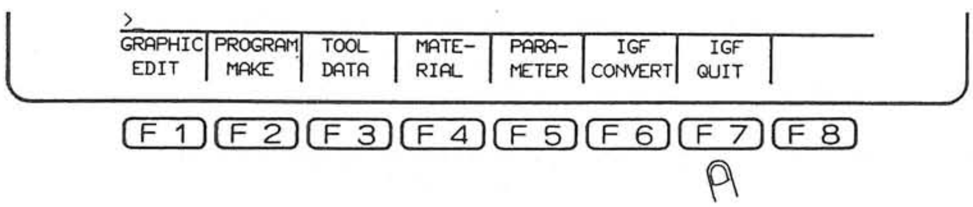
4) Press the function key [F2] (ORDER ↓) to display the *INTEGER PARAMETER* page.

```
PROG  OPERATION IGF      PARAMETER
 - 2-
                    *INTEGER PARAMETER*
 1 SEQUENCE NO. INCREMENT                       0
 2 MATERIAL CODE DIGITS                         2
 3 M41 MINIMUM RPM                             65
 4 M42 MINIMUM RPM                            205
 5 M43 MINIMUM RPM                              0
 6 M44 MINIMUM RPM                              0
 7 M41 MAXIMUM RPM                           1110
 8 M42 MAXIMUM RPM                           3500
 9 M43 MAXIMUM RPM                              0
10 M44 MAXIMUM RPM                              0
11 MAXIMUM SPINDLE RPM                       3500
12 ORDER PARAMETER                              5
13 TOOL DATA PARAMETER                          1
>PF
! OF
!0
!_
ORDER↑ | ORDER↓ |     |     |     |     | QUIT |
 F1      F2       F3    F4    F5    F6    F7     F8
```

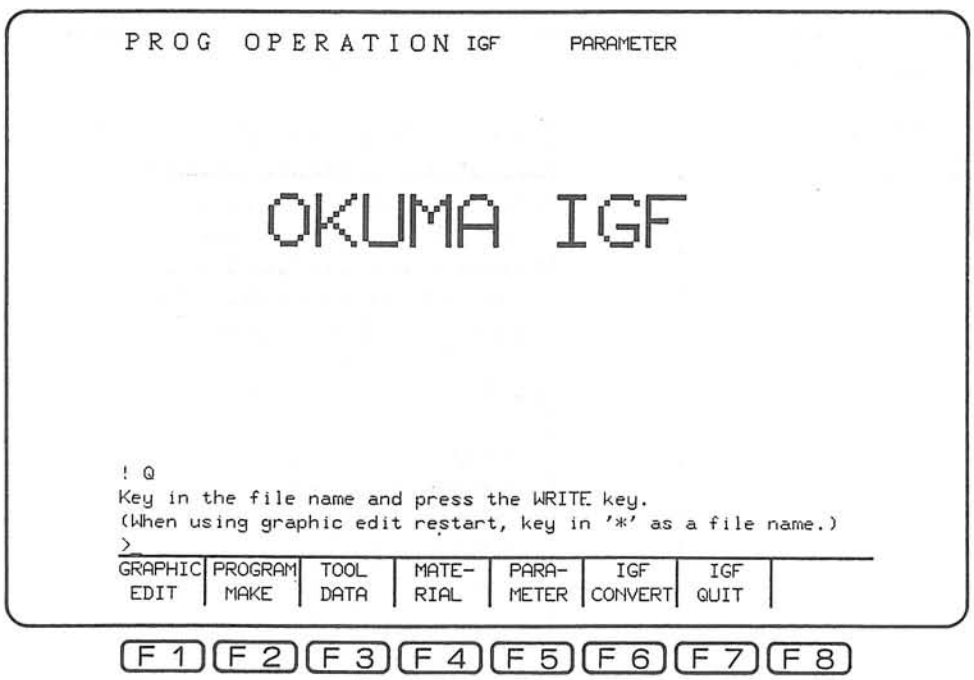
5) Locate the cursor on the SEQUENCE NO. INCREMENT and enter "Ø".



6) Press the function key [F7] (QUIT).



The display will be changed to the IGF start-up page.



	Initial Setting	Explanation	Reference*
DIMENSION PARAMETER			
1 OD CLEARANCE	1Ø		Section 9-3
2 ID CLEARANCE	3		
3 FACE CLEARANCE	3		
4 BLANK INSIDE LENGTH IL		Used for setting default of BLANK SIZE data entry parameters.	
5 BLANK INSIDE DIA. ID	Ø		
6 DIMENSION ZERO SHIFT	Ø		
7 PROGRAM ZERO SHIFT	Ø		
8 NOSE-R CANCEL TRAVEL	Ø.Ø1	Used for eliminating uncut portion in taper and arc cuts.	Section 4-1-2
9 CENTER LENGTH L2	8		
1Ø CENTER DIA. D2	36	The center dimensions are for LS3ØN.	
11 CENTER HOLE DIA. D3	5		
12 GROOVING CLEARANCE	Ø.1		
INTEGER PARAMETER			
1 SEQUENCE NO. INCREMENT	1Ø	1/1Ø of increment for assigning sequence numbers for each cutting process. When "Ø" is set, sequence numbers are assigned with consecutive numbers. "5" causes the first sequence of each process to be assigned as follows: NØØØØ 1st process NØØ5Ø 2nd process NØ1ØØ 3rd process NØ15Ø 4th process : : : :	

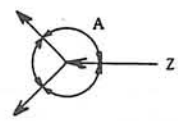
* Indicates the section number of the Application Manual (Publication No. 2476-E).

	Initial Setting	Explanation	Reference*
2 MATERIAL CODE DIGITS	2	Used for setting default of MATERIAL NO. (1 - 13).	
3 M41 MINIMUM RPM	5Ø	The lowest spindle speed (rpm) in each range	
4 M42 MINIMUM RPM	3ØØ		
5 M43 MINIMUM RPM	5ØØ		
6 M44 MINIMUM RPM	11ØØ		
7 M41 MAXIMUM RPM	25Ø	The highest spindle speed (rpm) in each range	
8 M42 MAXIMUM RPM	65Ø		
9 M43 MAXIMUM RPM	12ØØ		
1Ø M44 MAXIMUM RPM	22ØØ		
11 MAXIMUM SPINDLE RPM	2ØØØ	The allowable maximum spindle speed is set meeting the setup (G5Ø S2ØØØ).	
12 ORDER PARAMETER	5	Used for setting the time duration for automatic advance to the next process (page). Setting unit is 1ØØ msec. -1: Automatic advance not effective Ø - 1Ø: Automatic advance effective	
13 TOOL DATA PARAMETER	1	Used for selecting the data setting process for TOOL DATA setting: Ø: Designation of TOOL DATA NO advances the process to CONDITION data entry. 1: Designation of TOOL DATA No advances the process to TOOL DATA data entry. 2: Process advances to TOOL DATA data entry without the designation of TOOL DATA NO.	

	Initial Setting	Explanation	Reference*
14 TPI 15 NO. OF THREAD LEADS 16 NO. OF GROOVES	1 1 1	Used for setting default of corresponding CONDITION data entry parameters.	
17 FINISHING FEEDRATE	2	Used for setting default of finish symbols for CONDITION data entry.	
18 IGF FILE STORAGE	1	Used for designating whether IGF file is to be stored or not. Ø: Not stored 1: Stored	
19 THREAD INFEED PATTERN	73	Used for designating tool infeed pattern for thread cutting. 73: M73 mode in which depth of cut is controlled as D/2, D/4, D/8, D/8. 74: M74 mode in which depth of cut is maintained at a constant D. 75: M75 mode in which depth of cut is controlled as $\sqrt{2D}$, $\sqrt{3D}$.	Refer to the section covering compound fixed cycle in the Programming Manual.
2Ø THREAD CUTTING MODE	32	Used for designating cutting pattern of thread cutting operation. 32: M32 mode Straight infeed along thread face 33: M33 mode Zigzag infeed	

	Initial Setting	Explanation	Reference*
21 TOOL OFFSET CANCEL	Ø	Used for specifying whether the tool offset mode is canceled or not when the turret is returned to the turret index position. Ø: Cancel command (TO1ØØ, TG=Ø1, OG=Ø) output 1: Cancel command not output	
22 SPACE DELETION	Ø	Used for specifying whether spaces are output or not when making an NC program. Ø: Spaces output 1: Spaces not output 2: One space output	
23 COORDINATE AXIS LINE	Ø	Used for specifying the type of lines to draw coordinate axes. Ø: Solid line 1: Dotted line 2: Short dashes line 3: Long dashes line 4: Alternate short and long dashes line 5: Alternate long and two short dashes line 6: No axis display	
24 NOMINAL SIZE COMMAND (THREADING)	Ø	Used for selecting the thread diameter designation mode. Ø: Root diameter 1: Nominal diameter	

	Initial Setting	Explanation	Reference*
25 TOOL EDGE DATA CHECK MARGIN (ANGLE)	Ø	Used for setting margin angle for tool edge data. Ø° - 36Ø°	
26 TOOL EDGE DATA CHECK (ANGLE)	Ø		Section 6-9, "Tool Edge Shape Check" of Application Manual
27 TAILSTOCK BARRIER CHECK M-CODE OUTPUT	Ø		Section 7-4, (9), "Tool Interference Check and Barrier Check" of Application Manual
28 CHUCK BARRIER CHECK M-CODE OUTPUT	Ø		
29 AUTO COMBINED 4-AXIS PROGRAMMING (A+B)	1	Used for specifying that the simultaneous 4-axis control processes are automatically combined.	
3Ø A/B INTERCHANGEABLE PROGRAM	Ø	Used for specifying the status that the A-turret processes and the B-turret processes are replaced each other.	
31 TOOL INTERFERENCE CHECK M-CODE OUTPUT	Ø		Section 7-4, (9), "Tool Interference Check and Barrier Check" of Application Manual
32 NO. OF V-GROOVES	1		

	Initial Setting	Explanation	Reference*
PERCENT PARAMETER			
1 ROUGHING FEEDRATE ↖	15Ø	Factors (%) for determining the feedrate to be used when cutting is made along the shape in rough and copy turning operations. Factors are assigned for three different cutting directions such as ↖, ←, ↙ in reference to the main cutting direction.  ↖ $0° \leqq A < 135°$ ← $135° \leqq A \leqq 225°$ ↙ $225° < A < 36Ø°$	
2 ROUGHING FEEDRATE ←	1ØØ		
3 ROUGHING FEEDRATE ↙	1ØØ		
4 COPYING FEEDRATE ↖	15Ø		
5 COPYING FEEDRATE ←	1ØØ		
6 COPYING FEEDRATE ↙	1ØØ		
7 GROOVING SHIFT	9Ø	Tool shift factor in grooving operation. Shift amount is the tool width x set factor (%)	
8 GROOVING MID RETRACT	2ØØ	Factor used for determining tool retract and infeed amounts during grooving cycle. Retraction amount is the tool width x set factor (%).	
9 GROOVING END DWELL (RPM)	2ØØ	Dwell period at the end point of the grooving is specified in the number of spindle revolutions.	
1Ø DRILLING MID RETRACT	2ØØ	Factor used for determining drill retract and infeed amounts during drilling cycle. Retraction amount is the drill diameter x factor (%).	

	Initial Setting	Explanation	Reference*
11 THRU HOLE SLOW FEED CUT DEPTH	5Ø	Excess infeed amount for drilling a through hole is determined by this factor (%). Drill diameter x factor (%).	Section 4-4, "Drilling" of Application Manual
12 THRU HOLE SLOW FEEDRATE	5Ø	At the stage the drill is penetrating the workpiece, its feedrate is determined by this factor (%). Conventional drilling infeed rate x factor (%).	Section 4-4, "Drilling" of Application Manual
13 DRILLING END DWELL (RPM)	2ØØ	Dwell period at the end point of the drilling is specified in the number of spindle revolutions.	
14 CENTERING END DWELL (RPM)	2ØØ	Dwell period at the end point of the centering is specified in the number of spindle revolutions.	
15 RECESSING END DWELL (RPM)	2ØØ	Dwell period at the end point of the recessing is specified in the number of spindle revolutions.	
16 THREAD CHAMFER	Ø	Factor (%) used for determining the chamfering amount in thread cutting operation. Chamfering amount is F/J x factor (%).	
17 HEIGHT OF SCREW THREAD (EXTERNAL)	65	Factors (%) used for designating the default of HEIGHT.	
18 HEIGHT OF SCREW THREAD (INTERNAL)	5Ø	HEIGHT = LEAD/(TPI x NO. OF THREAD LEADS) x FACTOR(%) /1ØØ	
19 HEIGHT OF SCREW THREAD (END FACE)	87		
2Ø 4-AXIS CUTTING CONDITIONS (RPM)	12Ø	Setting range: Ø - 2ØØ%	
21 4-AXIS CUTTING CONDITIONS (CUT DEPTH)	7Ø	Setting range: Ø - 1ØØ%	

	Initial Setting	Explanation	Reference*
22 4-AXIS CUTTING CONDITIONS (DRILL FEED)	7Ø	Setting range: Ø - 1ØØ%	
23 4-AXIS CUTTING CONDITIONS (GROOVE FEED)	8Ø	Setting range: Ø - 1ØØ%	
24 SPINDLE OVERRIDE FOR SIMULATION	1ØØ	Setting range: Ø - 2ØØ%	
25 FEEDRATE OVERRIDE FOR SIMULATION	1ØØ	Setting range: Ø - 2ØØ%	
4Ø BALANCED CUTTING DEPTH	2ØØ		Section 4-5, "Balance Cut" of Application Manual
41 BALANCED CUTTING FEEDRATE	1ØØ		
Parameters Used for Controlling Spindle *SPINDLE ROT*		Used for designating the default of CW/CCW parameter of CONDITION data setting process. Ø: CW 1: CCW	
CONST SPEED		Used for designating the default of constant speed cutting mode selection. Ø: Constant speed cutting OFF 1: Constant speed cutting ON	
Parameters Used for Controlling Tool Change *RETRACT* *APPROACH*		Used for designating the retraction mode to the turret indexing position and the approaching mode to the circle reference point. Ø: One axis 1: Two axes 2: Gang tool plate	

	Initial Setting	Explanation	Reference*
Material Parameters			
DRILLING (CARBIDE) (HSS)		Used for designating the step feed mode in drilling. Ø: No step feed 1: Step feed	
GROOVING		Used for designating the step feed mode in grooving. Ø: No step feed 1: Step feed	
COOLANT		Used for designating the default of coolant on/off state for individual cutting processes. Ø: Coolant OFF 1: Coolant ON	
BAR FEEDER/PULLER PARAMETERS			
1 BAR FEEDER/PULLER OPERATION PATTERN	Ø	Used for designating the operation pattern of the bar feeder/puller.	Section 12, "Bar Feeder and Bar Puller Function" of Application Manual
2 BAR FEEDER/PULLER PROGRAM	Ø	Used for designating whether or not bar feeder (puller) controlling M codes are output when making a program.	
3 CUTTING SPEED	8Ø	Setting range: 1 - 32ØØ (mm/rev)	
4 CUTTING FEED	Ø.1	Setting range: Ø.ØØ1 - 9999.999 (mm/min)	
5 CUTTING SPINDLE	1	Used for designating the spindle rotation direction for top-cut operation.	
6 CUTTING COOLANT	1	Used for designating coolant ON or OFF for top-cut operation	
7 STOP STARTING POSITION WITH TOP CUTTING	Ø	Setting range: Ø.ØØ1 - 9999.999 (mm)	

	Initial Setting	Explanation	Reference*
8 STOP STARTING POSITION WITH LINK FEED		Setting range: Ø.ØØ1 - 9999.999 (mm)	Section 12, "Bar Feeder and Bar Puller Function" of Application Manual
9 AUTO STOP LINK FEEDRATE		Setting range: Ø.ØØ1 - 9999.999 (mm/min)	
1Ø STOCK OUT LENGTH (FL)		Setting range: Ø.ØØ1 - 9999.999 (mm)	
11 BAR PULLER STARTING POSITION (PS)		Setting range: Ø.ØØ1 - 9999.999 (mm)	
12 BAR PULLER GRIPPING DISTANCE (D)		Setting range: Ø.ØØ1 - 9999.999 (mm)	
13 BAR PULLER REACHING SPEED		Setting range: Ø.ØØ1 - 9999.999 (mm/min)	
14 BAR PULLER DRAW SPEED		Setting range: Ø.ØØ1 - 9999.999 (mm/min)	

SECTION 5 GRAPHIC EDIT FUNCTION

This section covers the graphic editing processes, in which a series of NC programming beginning with material designation up to the conversion of the NC data into an NC program. In this process, data is entered in the interactive mode for material, blank shape, setup data (chuck and tailstock data) and process designation.

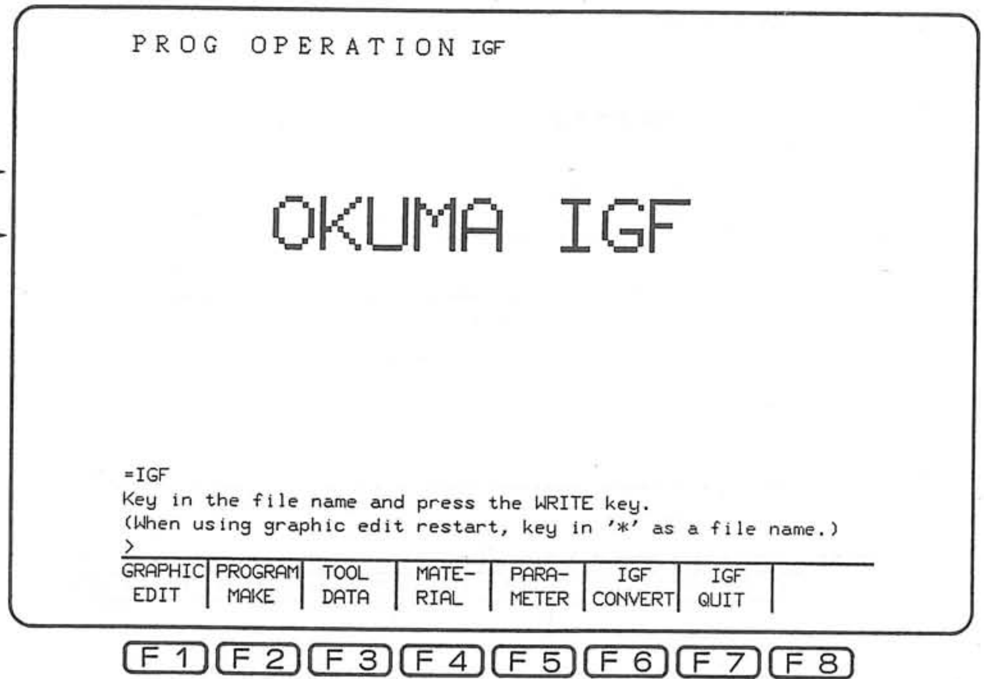
During the graphic edit process, it is possible to check the program through the simulation for optimizing the machining through the change of machining sequences and cutting conditions. The result of optimization can calculated in the form of cycle time and displayed on the CRT.

Main Steps	Contents
Material designation	Select the material from the registered 13 types of materials from the material table.
Blank shape	Enter the data to define the blank material. This includes blank length and blank diameter(s). This designation is made both for outside and inside diameters of a blank.
Chuck and tailstock	Enter the dimensions of chuck jaws and tailstock center and also the availability of the tailstock center.
Process designation	Enter the data necessary for process creation, copy and deletion operations.
Process table	Simulation of the turning operation, change of turning sequences and display of cycle times for individual processes are possible.
Program making	NC program is created from the IGF data.

Graphic Edit Processes

1) After selecting the PROGRAM OPERATION mode, press the function key [F7] (IGF) to display the IGF start-up page.

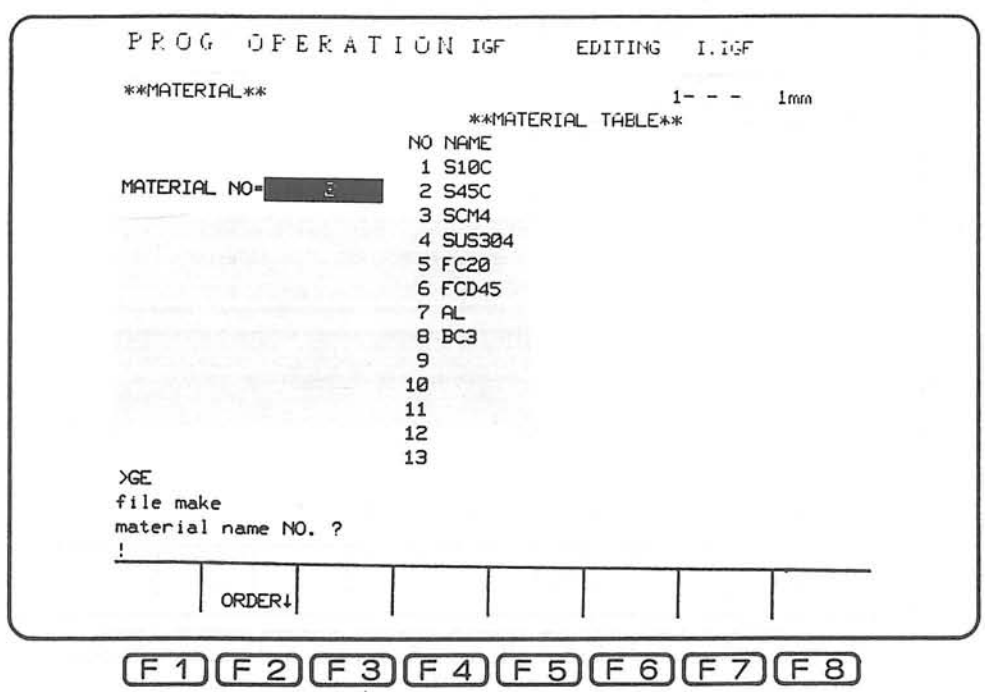
Press the function key [F1] (GRAPHIC EDIT).



[WRITE]

2) Designate the material. The material table is called out from the MATERIAL data file.

[F7]
(QUIT)

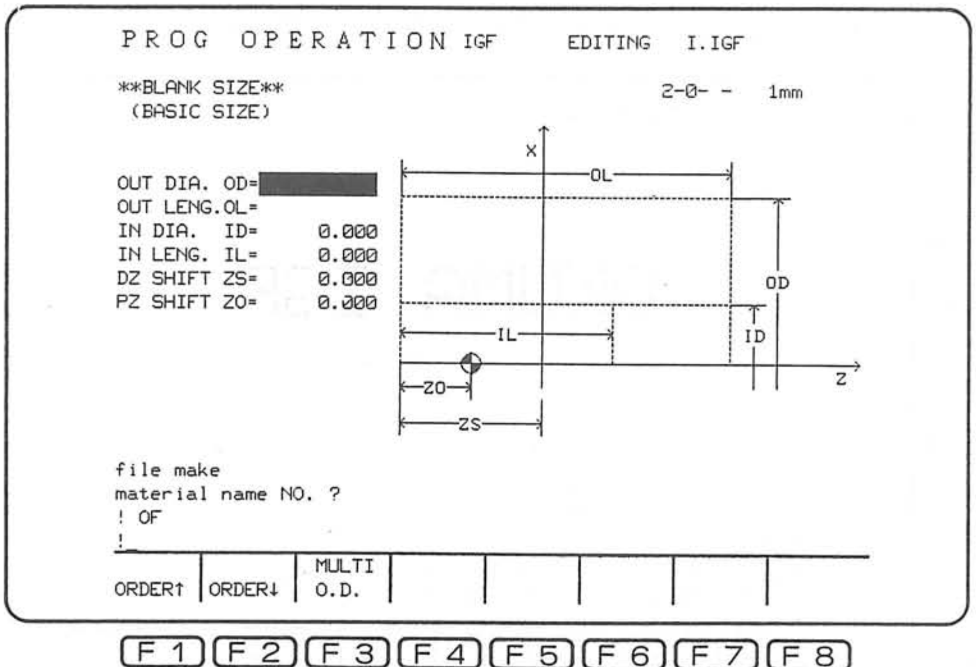


[F1] (ORDER ↑) [F2] (ORDER ↓)

[F1] (ORDER ↑) [F2] (ORDER ↓)

3) Enter the blank size and programming zero.

[F7] (QUIT)

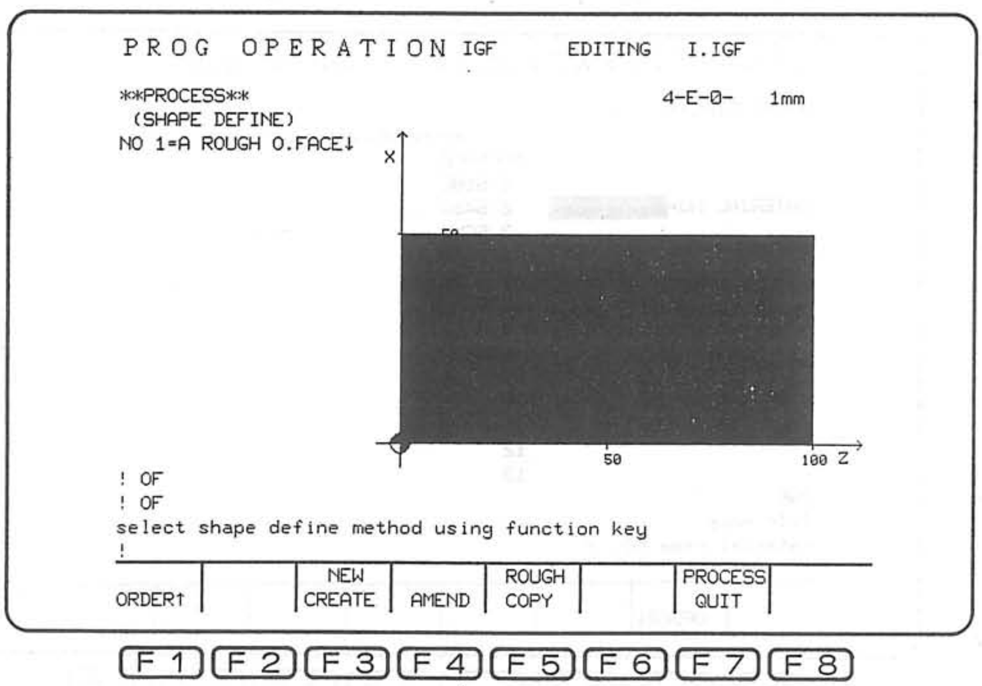


[F1] (ORDER ↑) [F2] (ORDER ↓)

4) Designate the turning processes*.

Enter the shape for individual processes such as rough turning, finish turning, thread cutting and grooving.

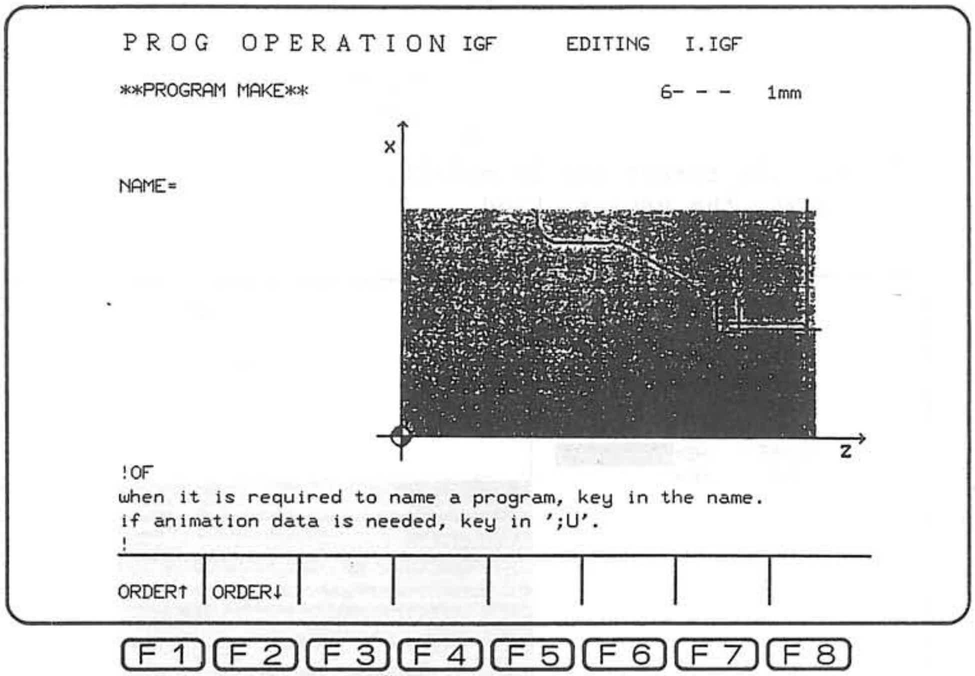
[F7] (QUIT)



[F2] (ORDER ↓)

[F2] (ORDER ↓)

5) Enter the program name.



* For the process designation, refer to the following page. Note that the operations not covered in this manual such as multi-diameter blank, setup data and simulation (process test) are explained in details in the Application Manual (Publication No. 2476-E).

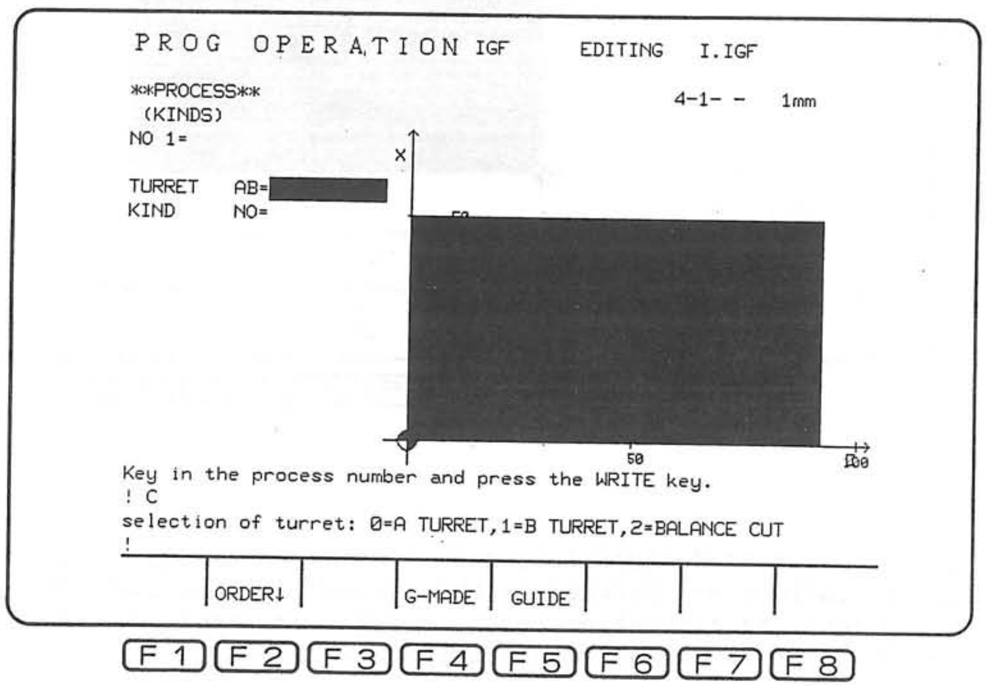
Process Designation Procedure

1) Process Designation

To register a new process, press the function key [F3] (CREATE).

[WRITE]
(Refer to Note.)

2) Select the turret for 2S models.
Designate the process kind.



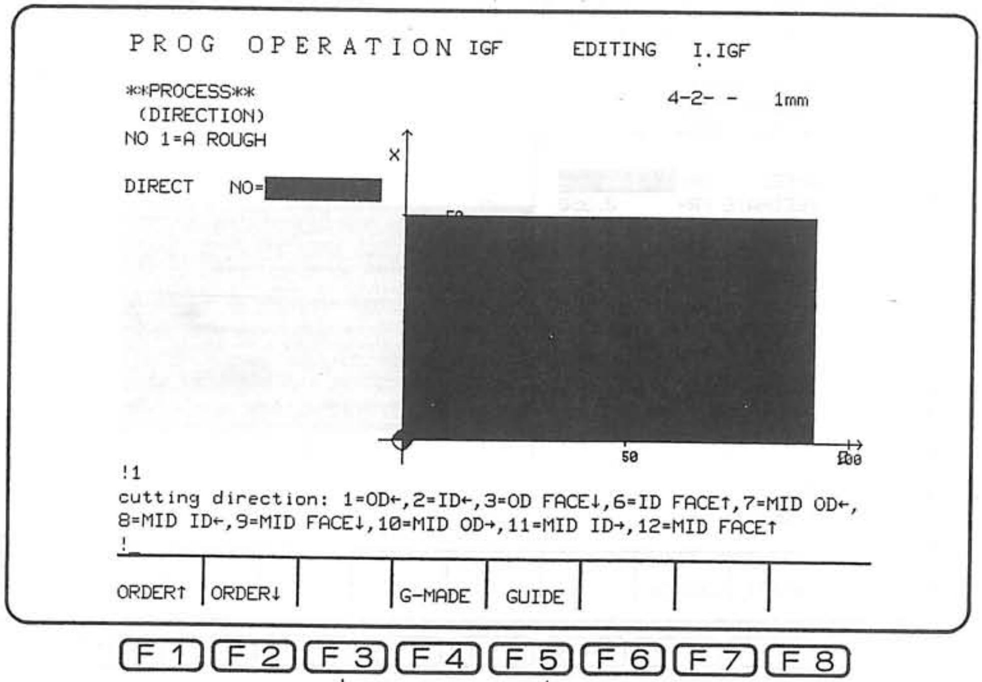
[F1]
(ORDER ↑)

[F2]
(ORDER ↓)

[WRITE]

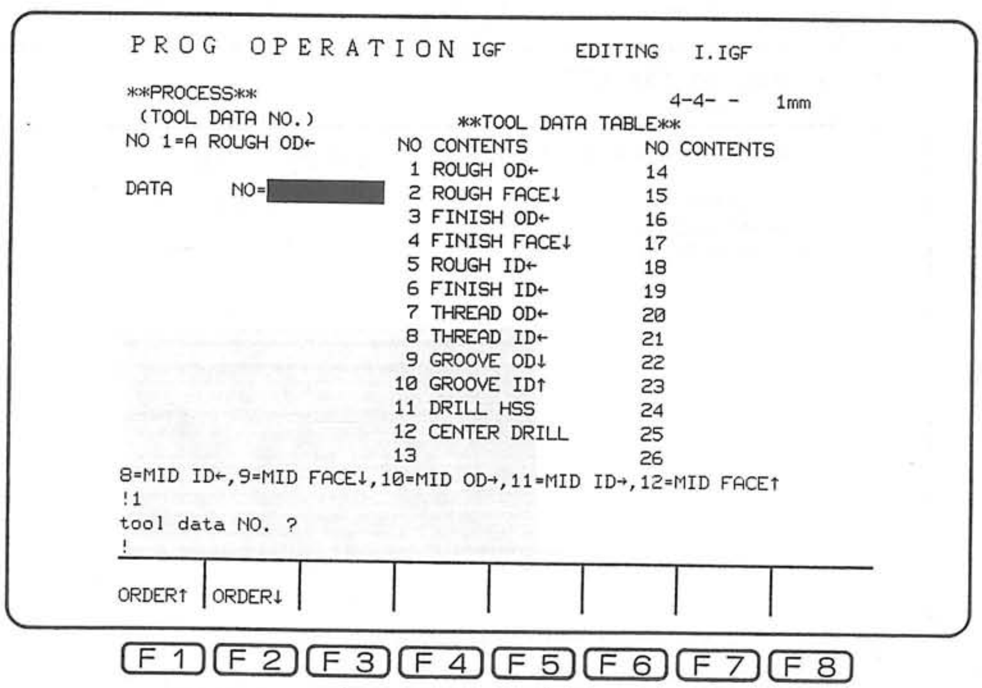
[F1] (ORDER ↑) | [F2] (ORDER ↓) | [WRITE]

3) Designate the cutting direction.



[F1] (ORDER ↑) | [F2] (ORDER ↓) | [WRITE]

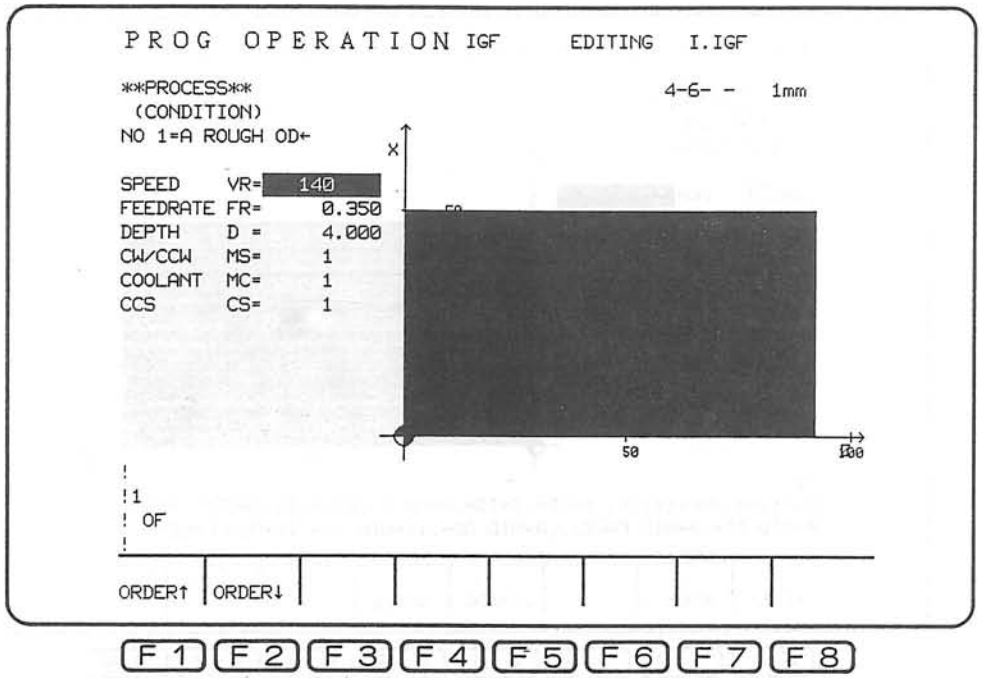
4) Designate the cutting tool.



[F1] (ORDER ↑) | [F2] (ORDER ↓) | [WRITE]

[F1] (ORDER ↑) | [F2] (ORDER ↓) | [WRITE]

5) Designate the cutting conditions.

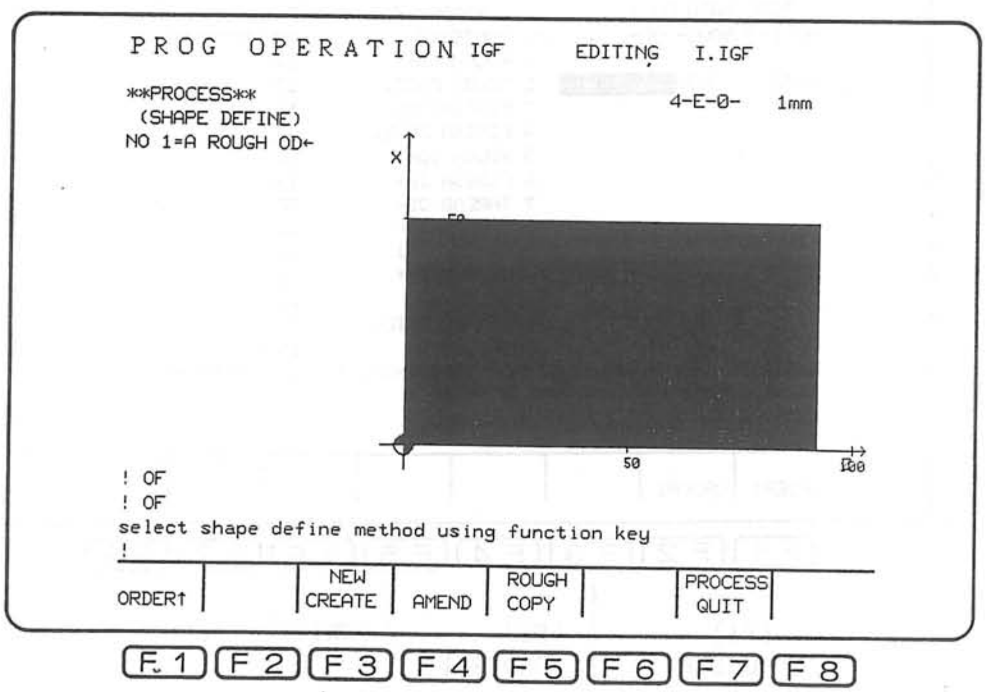


[F1] (ORDER ↑) | [F2] (ORDER ↓) | [WRITE]

6) Shape Definition

Designate blank shape and, then, enter the reference point of the cycle, start point of the shape element, etc. in accordance with the prompt on the CRT.

[F7] (QUIT)



Note: The control first checks whether or not there is enough empty space for registering one process data when process creation is attempted during process designation. If not enough space is remaining, the following alarm message is displayed.

WARNING: the bubble (floppy-disk) lacks free area.
before continuing this IGF operation, arrange
the bubble (floppy-disk).

This alarm message is displayed when there is not enough space for the maximum number of elements which is 48, during the arbitrary shape definition process. That is, it is possible that some processes can be stored in spite of the alarm message display depending on the size of the process.

Ordinarily, what is to be done when this warning message is displayed to abort process creation and proceed to program creation step and then store the processes made in the IGF file and escape IGF mode. After that, erase unnecessary files in the bubble memory to obtain more empty space in the bubble memory and return to the IGF to continue process creation.

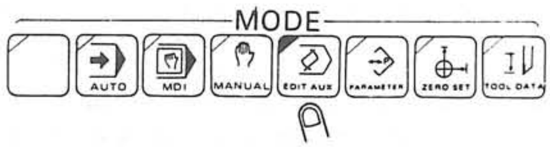
It is also possible to interrupt the process creation process using the IGF temporary quit and recovery function and then resume the operation after increasing available space in the bubble memory. For the details of the function, refer to Section 1, "IGF Temporary Quit and Recovery Function" of Application Manual (Publication No. 2476-E).

SECTION 6 PROGRAMMING

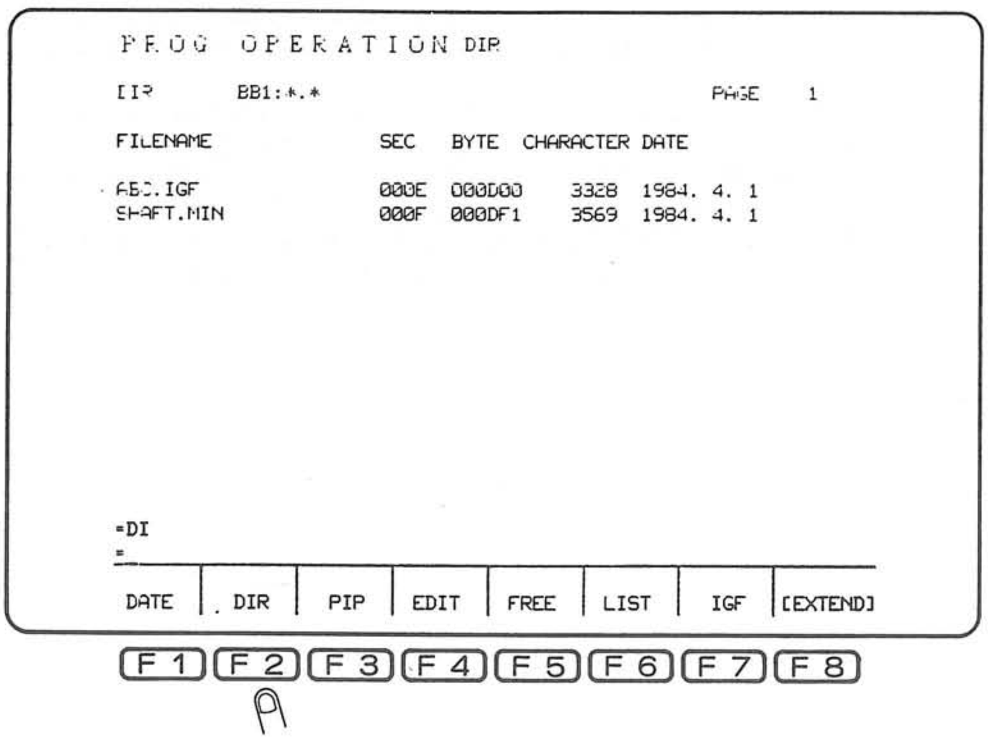
This section explains the procedure to create an NC program file using the data already registered in the IGF file without carrying out the graphic edit operation. The NC program file can be assigned with a program name.

Programming Procedure:

1) Select the PROGRAM OPERATION mode by pressing the EDIT AUX. key.

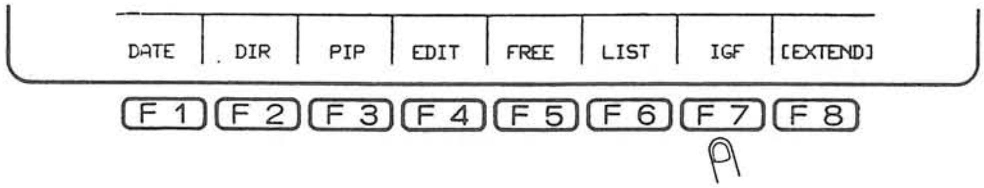


2) Press the function key [F2] (DIR) and then press the WRITE key. The CRT will display the names of registered files.

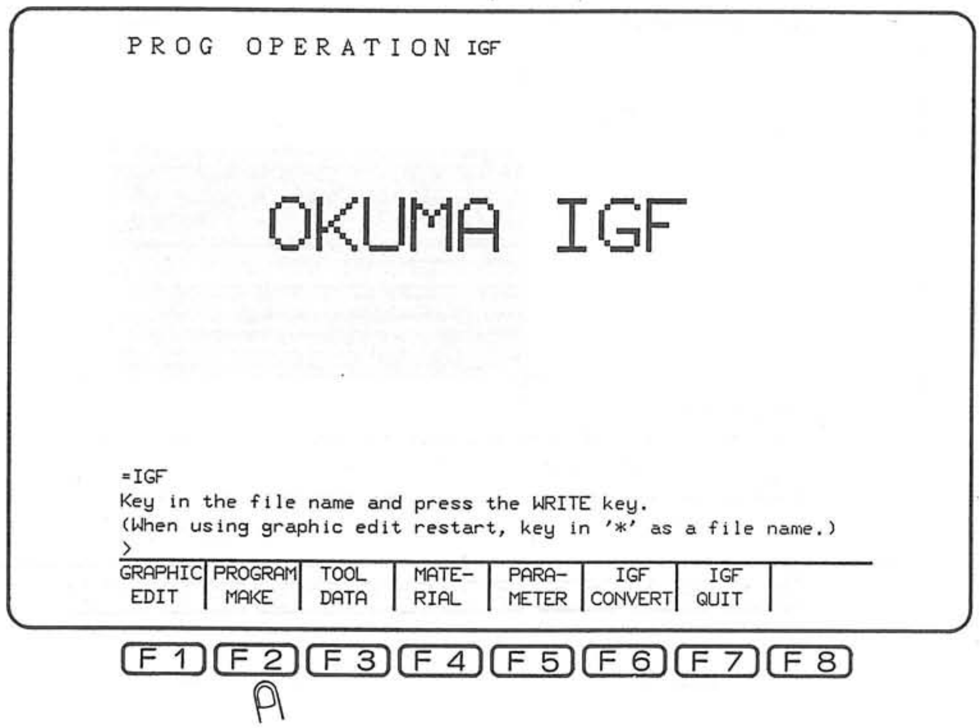


Select the necessary IGF file for programming and assign the program name.

3) Press the function key [F7] (IGF).



4) The page is changed to the IGF start up page. Press the function key [F2] (PROGRAM MAKE).



The prompt "=PM" should be displayed on the CRT.

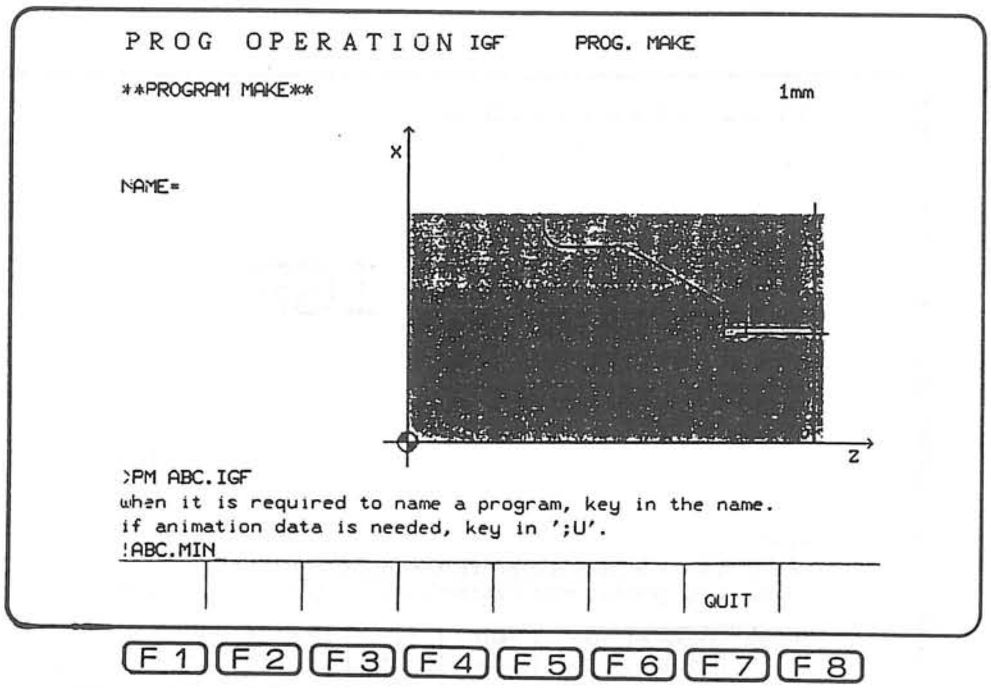
5) Key in the IGF file name selected following the prompt "PM".

>PM ABC.IGF

Then, press the WRITE key.

In this file name keying-in operation, extension .IGF may be omitted.

>PM ABC [WRITE]



6) Enter the required NC program file name.

! ABC.MIN [WRITE]

The extension .MIN is also omissible.

! ABC [WRITE]

When a program name is used, key-in the program name following the file name with a comma between the program and file names.

! ABC.MIN, O1ØØØ [WRITE]

In this case, omission of extension .MIN is also acceptable.

! ABC, O1ØØØ [WRITE].

Note: For subprograms (extension .SUB) and system subprograms (extension .SSB), a subprogram name (or number) must be assigned in addition to the program file name.

Example: A.SUB,O1ØØØ [WRITE]

Example of Operations

a) Creation of A.MIN from I.IGF

Key Operations	Simplified Entry
>PM I.IGF [WRITE] !A.MIN [WRITE]	>PM [WRITE] ! [WRITE]

b) Creation of ABC.MIN from Q.IGF

Key Operations	Simplified Entry
>PM Q.IGF [WRITE] !ABC.MIN [WRITE]	>PM Q [WRITE] ! ABC [WRITE]

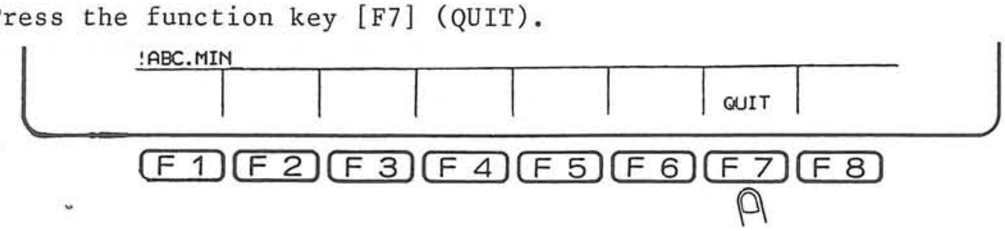
c) Creation of ABC.SUB from Q.IGF

Key Operations	Simplified Entry
>PM Q.IGF [WRITE] !ABC.SUB,OABC [WRITE]	>PM Q [WRITE] ! ABC.SUB,OABC [WRITE]

d) Creation of D.SSB from I.IGF

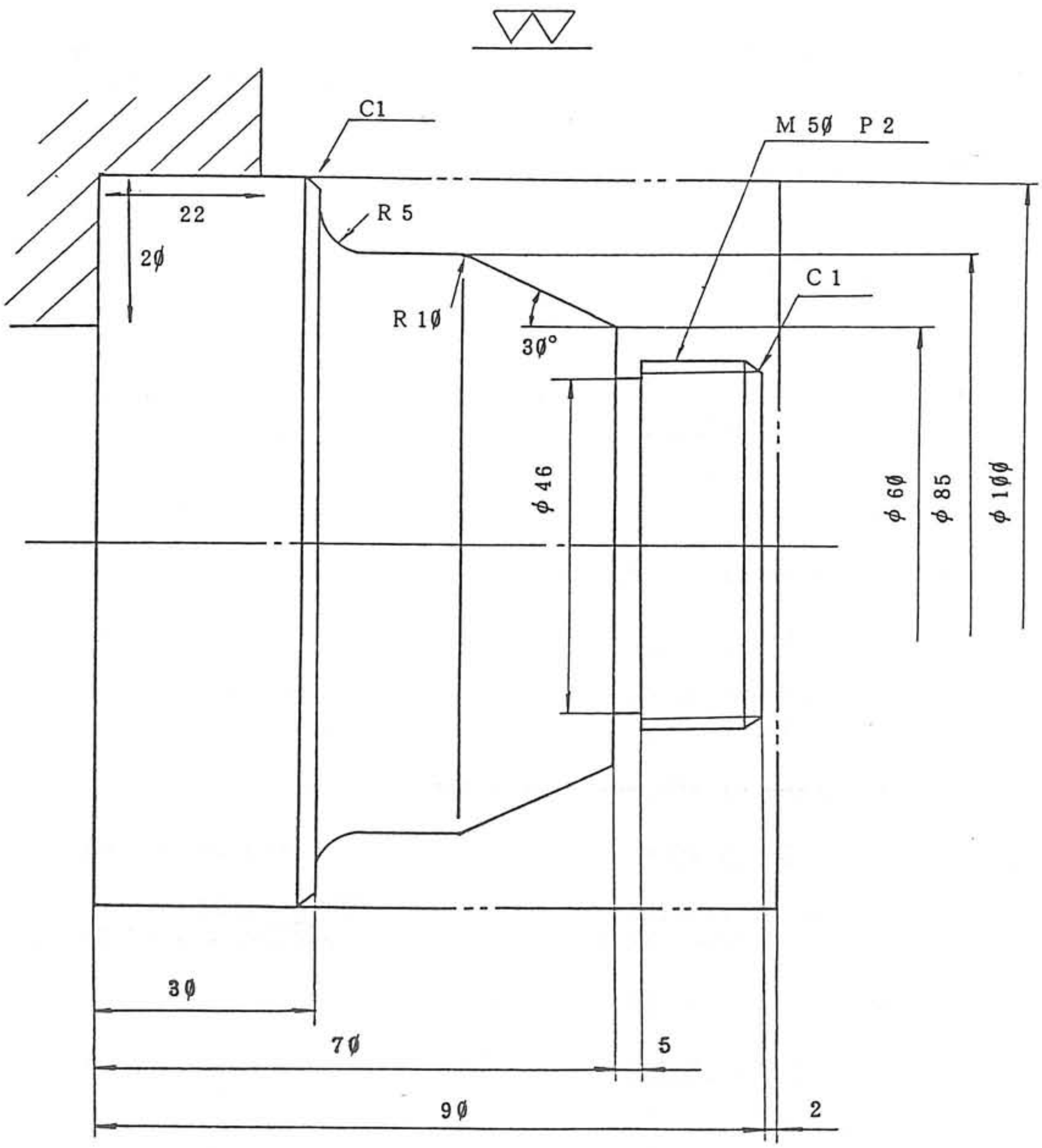
Key Operations	Simplified Entry
>PM I.IGF [WRITE] !D.SSB,O1ØØØ [WRITE]	>PM [WRITE] ! D.SSB,O1ØØØ [WRITE]

7) Press the function key [F7] (QUIT).



SECTION 7 OPERATION PROCEDURES

This section provides an example of IGF system operation using as a sample the workpiece illustrated below. These instructions will be more meaningful if you actually use them to operate your system, instead of merely reading them here.



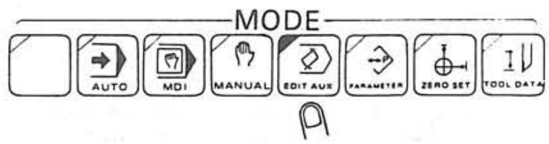
Material: S45C (JIS)
Blank : Bar (φ1ØØ x 92 mm)

Fig. 7-1

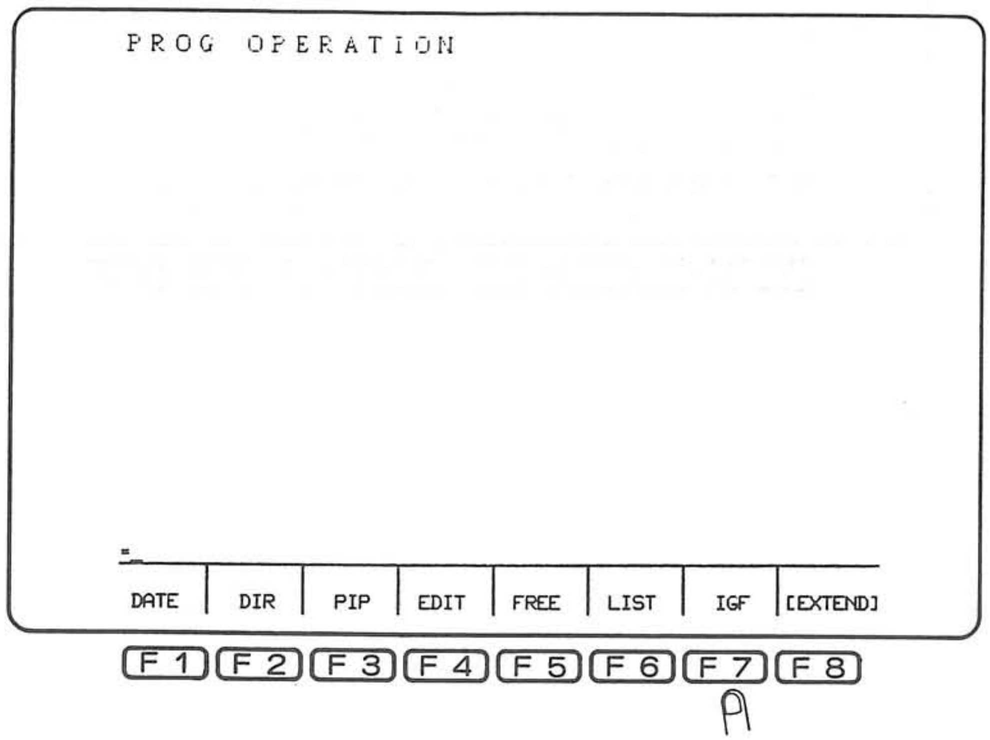
7-1. STARTING-UP IGF SYSTEM

First select the IGF mode.

1) Press the EDIT AUX. key.

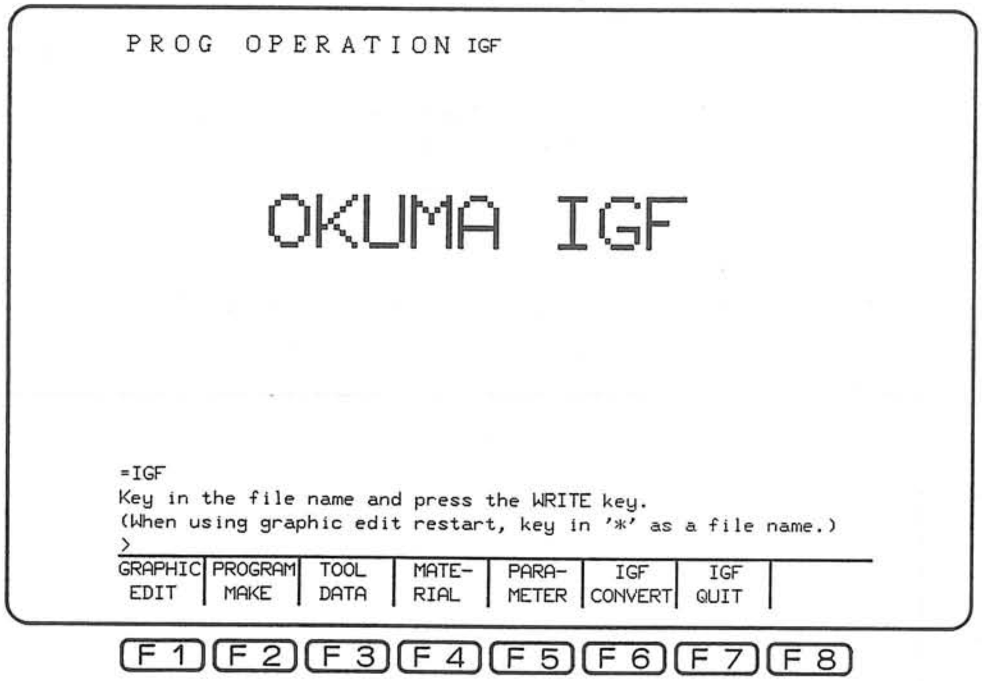


2) The CRT display will be changed as shown below.



3) Press the function key [F7] (IGF).

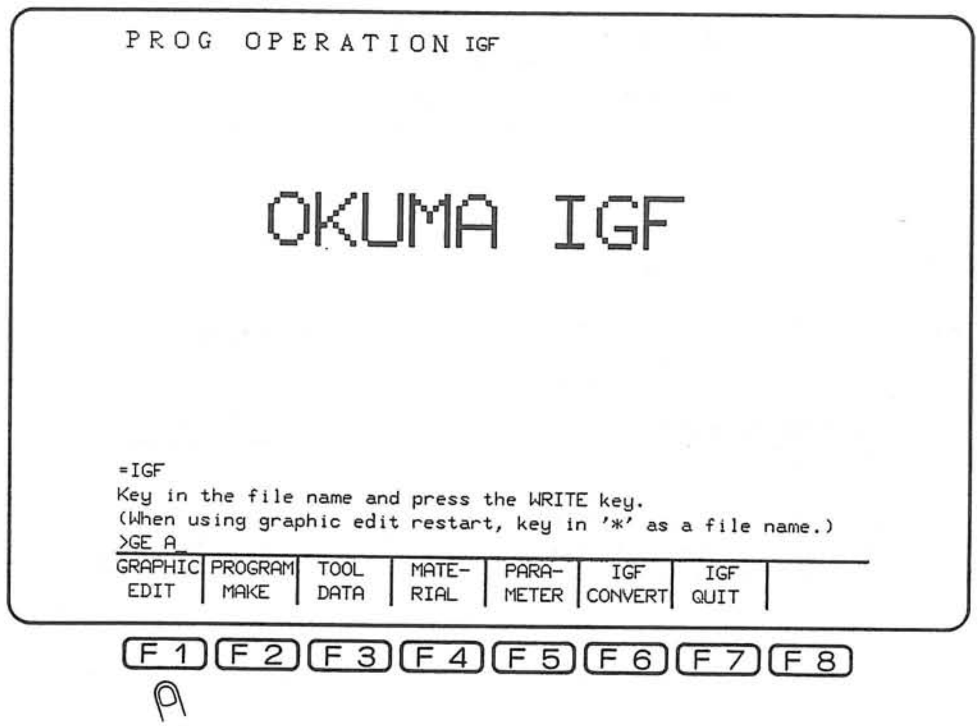
4) This selects the IGF mode. The CRT display will be changed as follows (IGF start-up page).



7-2. TURNING ON GRAPHIC EDIT FUNCTION

For programming a part, select the GRAPHIC EDIT function.

1) Press the function key [F1] (GRAPHIC EDIT) from the IGF start-up page.

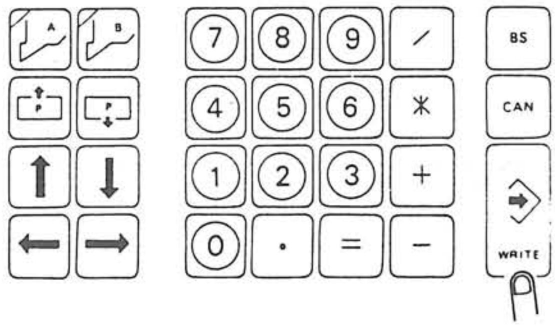


2) When multiple set of IGF data is required, enter the IGF file names from the keyboard.

   Example: For assigning the IGF file name "A.IGF", touch alphabet key A.

   If this IGF file name entry step is omitted, the file is automatically assigned with "I.IGF".

3) Press the WRITE key.



Reference:

a) Generating a new file >GE A [WRITE]

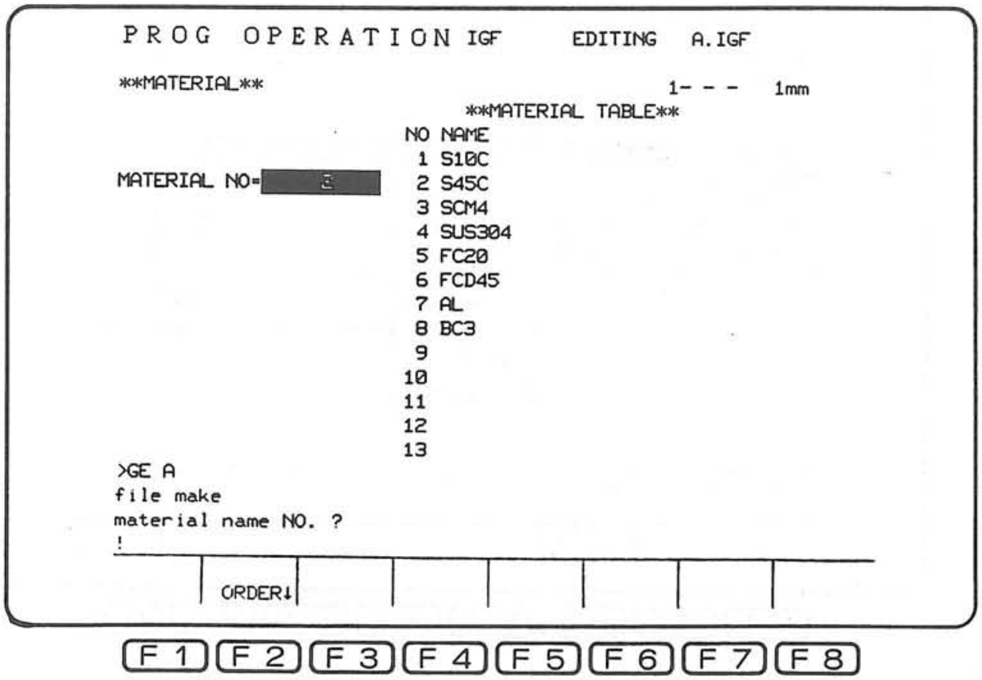
b) Editing a file already registered with file name A.IGF >GE A [WRITE]

c) Generating a file B.IGF by editing the IGF data in IGF file A.IGF >GE A,B [WRITE]

7-3. MATERIAL DESIGNATION

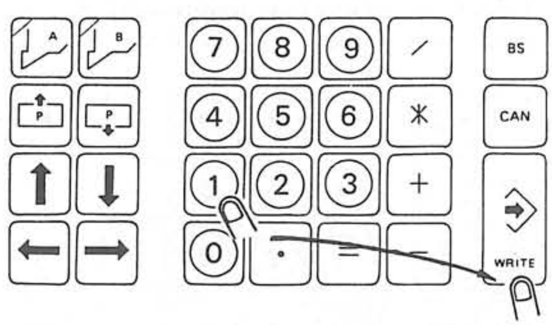
Select the blank material from the MATERIAL TABLE. The CRT screen displays the default value "2" (S45C). Therefore, special operations to select blank material is not necessary if S45C is used. For other blank materials, select the material from the menu.

1) Display the **MATERIAL** page.

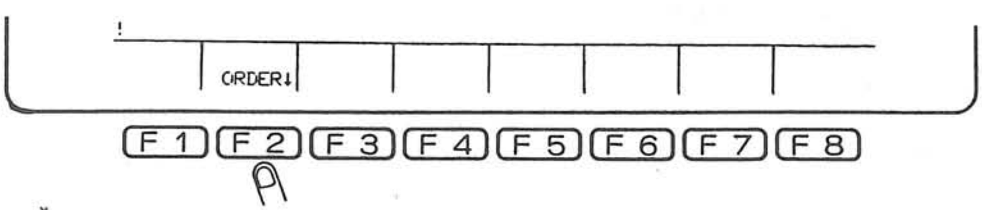


For this example, no key operation is required since material used is S45C.

For S1ØC, enter "1" for MATERIAL NO.



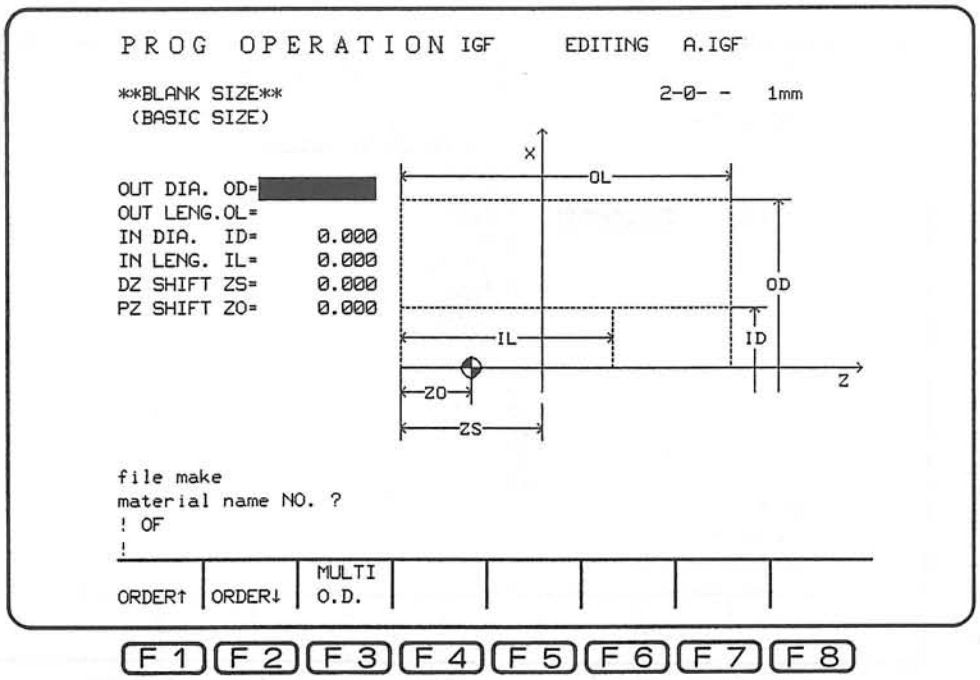
2) After making sure that "2" is displayed on the CRT, press the function key [F2] (ORDER↓).



7-4. BLANK SIZE ENTRY

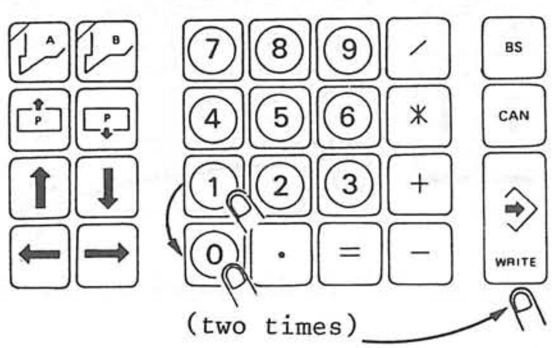
Enter blank diameter and blank length.

Make sure that the **BLANK SHAPE** page is displayed.



1) Enter "1ØØ" for blank outside diameter (OUT DIA. OD=).

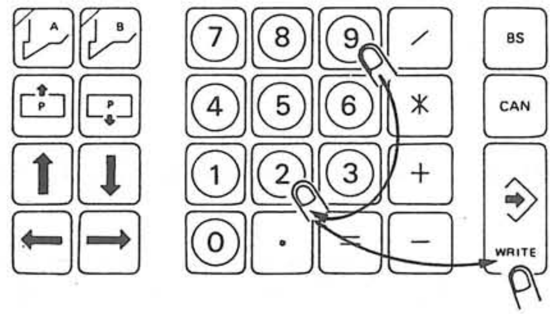
   This is accomplished by pressing numerical keys "1", "Ø" and "Ø" in this order and then pressing the WRITE keys.



Entry of "1ØØ"

2) Enter "92" for blank out length (OUT LENG. OL=).

This is accomplished by pressing numerical keys "9" and "2" and then pressing the WRITE key.

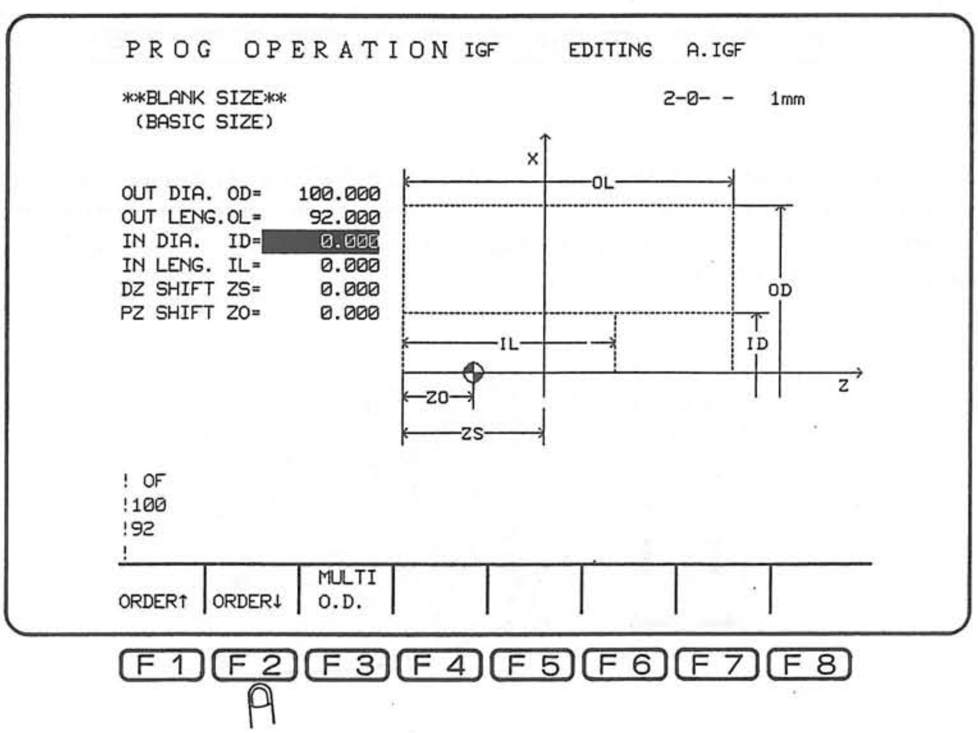


Entry of "92"

For other parameters, DZ SHIFT ZS (reference point) and PZ SHIFT ZO (programming zero), data entry may be omitted because both the reference point (ZS) and the programming zero (ZO) are taken at the left end of the workpiece and that they are taken as the reference (ZØ) for longitudinal dimensioning of a workpiece.

For workpieces with multiple diameters, refer to the Application Manual (Publication No. 2476-E).

3) Press the function key [F2] (ORDER ↓) to advance to the **CHUCK & CENTER** page.

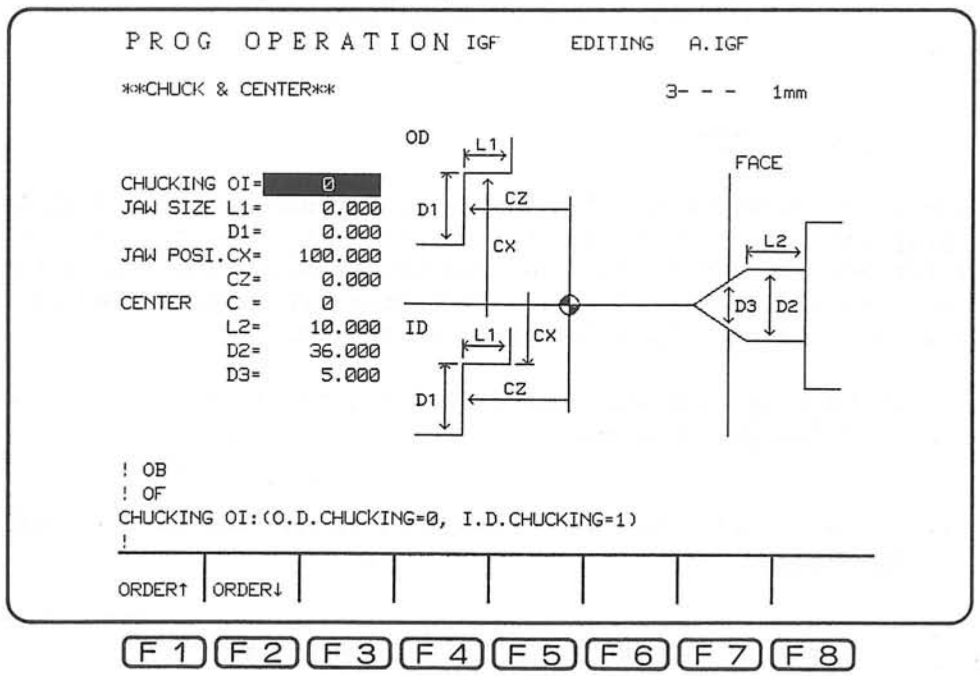


7-5. CHUCK AND CENTER DATA ENTRY

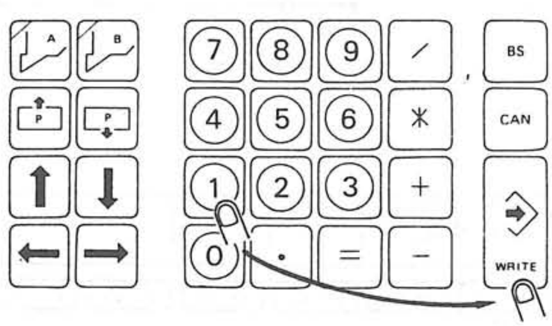
From the CHUCK & CENTER page, dimensions of the chuck jaws and the tailstock spindle center are entered referring to the graphic display of them.

For the machine equipped with the tailstock spindle center, whether the M codes for center barrier check are to be output or not is determined by the IGF parameter.

Make sure that the display is the **CHUCK & CENTER** page.

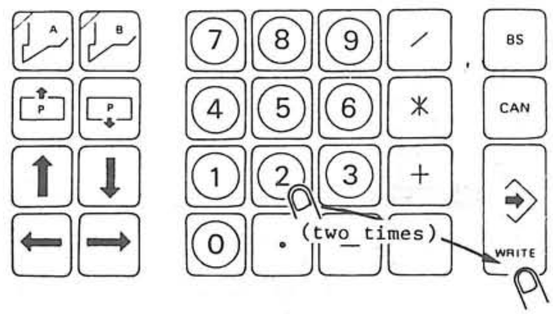


1) For the parameter CHUCKING OI, default "Ø", selecting the OD gripping, is set. For this example, no key operations are required for the selection of gripping direction since OD gripping is used. For ID gripping, enter "1".



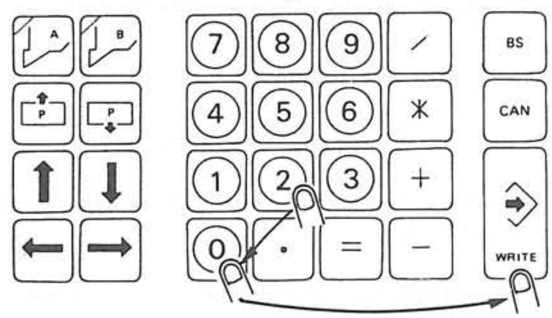
2) Enter "22 mm" for the parameter JAW SIZE L1.

This is accomplished by pressing the numerical key "2" two times and then pressing the WRITE key.



3) Enter "2Ø mm" for the parameter JAW SIZE D1.

This is accomplished by pressing the numerical keys "2" and "Ø" and then pressing the WRITE key.

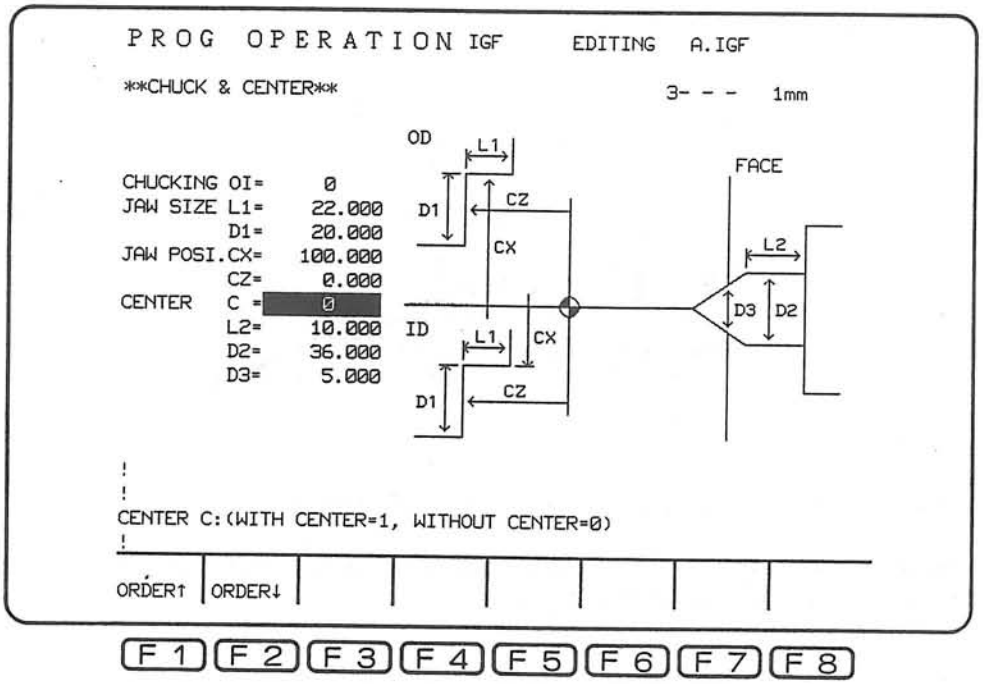


4) For JAW POSI. CX, the blank diameter keyed-in in the blank designation step is automatically set. Therefore, setting is not required for OD gripping. However, for ID gripping, setting of JAW POSI. CX is necessary.

5) For JAW POSI. CZ, default "Ø" is set. Set the position data in reference to the programming zero.

No key operations are required for this example.

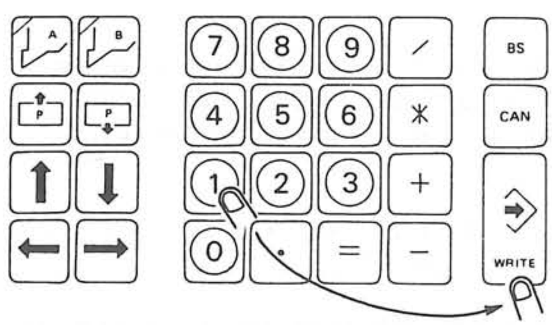
The display will be as indicated below.



6) For CENTER C, default "Ø (WITHOUT CENTER)" is set.

In this example, since the tailstock spindle center is not used, no key operations are required.

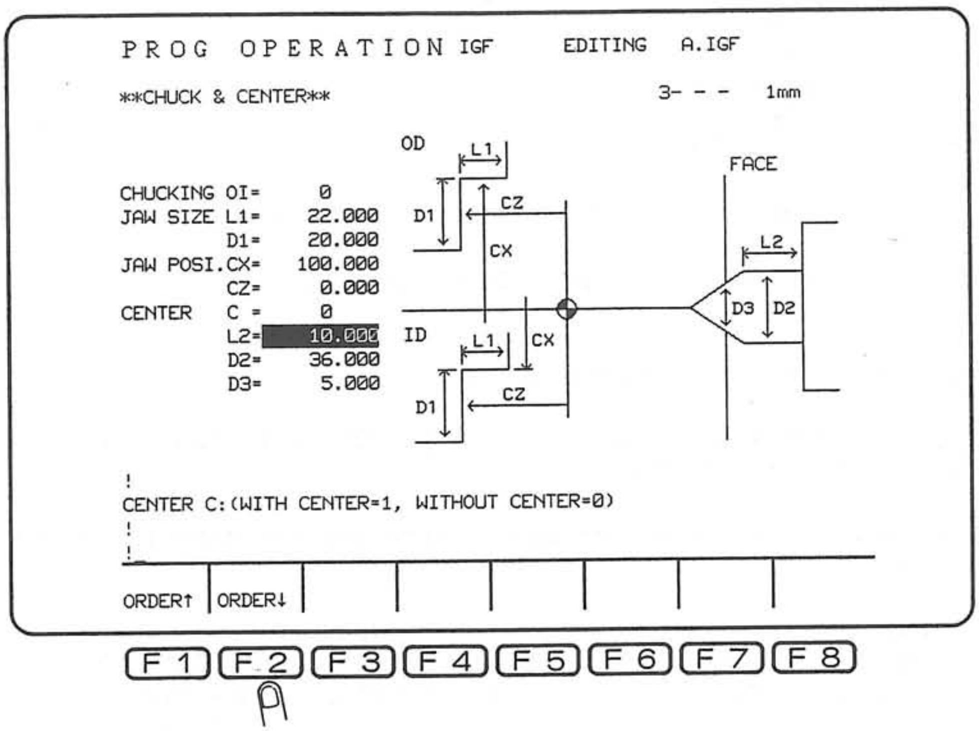
When the tailstock spindle center is to be used, enter "1".



Reference: If "1 (WITH CENTER)" is selected, it is necessary to designate whether the M code for the center barrier check is to be output or not by the IGF parameter.

7) For other parameters of the tailstock spindle center, data entry is not required.

Press the function key [F2] (ORDER ↓) to advance to the next process, **PROCESS**.



7-6. PROCESS DESIGNATION

For turning a sample workpiece illustrated in Fig. 7-1, the following six processes are required from the standpoint of tool types and shapes to be turned.

NO 1 = ROUGH O.FACE ↓
NO 2 = ROUGH OD ←
NO 3 = FIN. O.FACE ↓
NO 4 = FIN. OD ←
NO 5 = GROOVE OD ↓
NO 6 = THREAD OD ↓

The processes are designated in the sequences above.

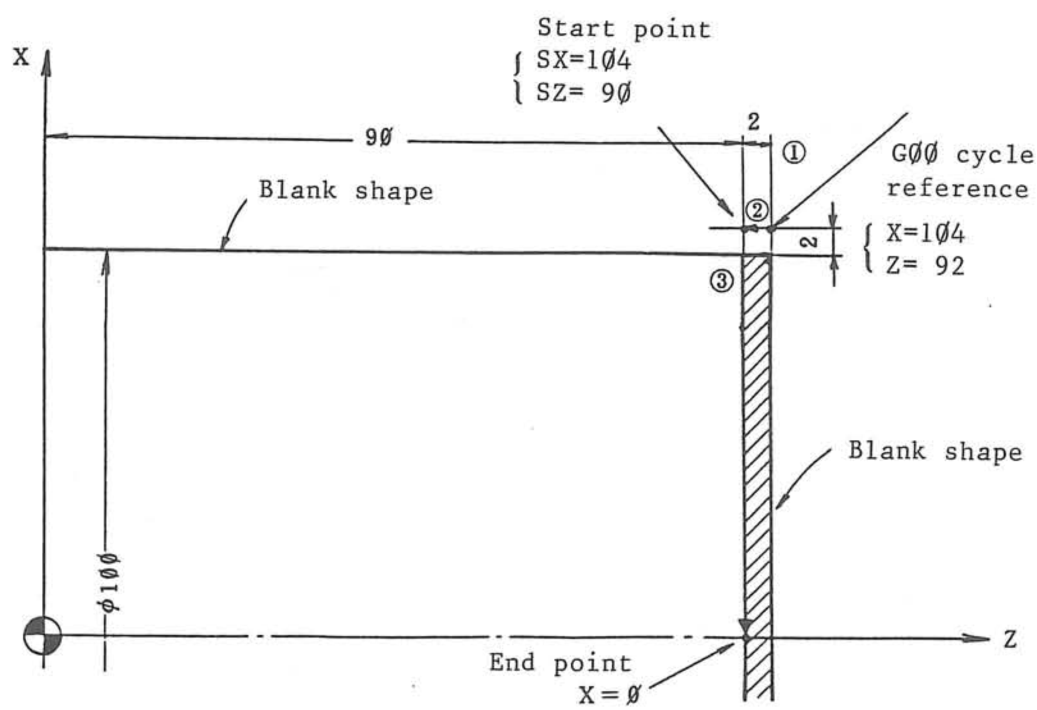
The IGF system can accept up to 24 processes.

7-6-1. Registering Rough OD Face Process

The ROUGH O.FACE process removes the portion indicated by hatching lines illustrated in Fig. 7-6-1.

This process can be registered by entering the data in the following sequence.

1) PROCESS
2) PROCESS (KINDS)
3) PROCESS (DIRECTION)
4) PROCESS (TOOL DATA NO)
5) PROCESS (CONDITION)
6) PROCESS (SHAPE DEFINE)



PROCESS NO. 1 ROUGH O.FACE ↓

Fig. 7-6-1 Rough OD Facing Process

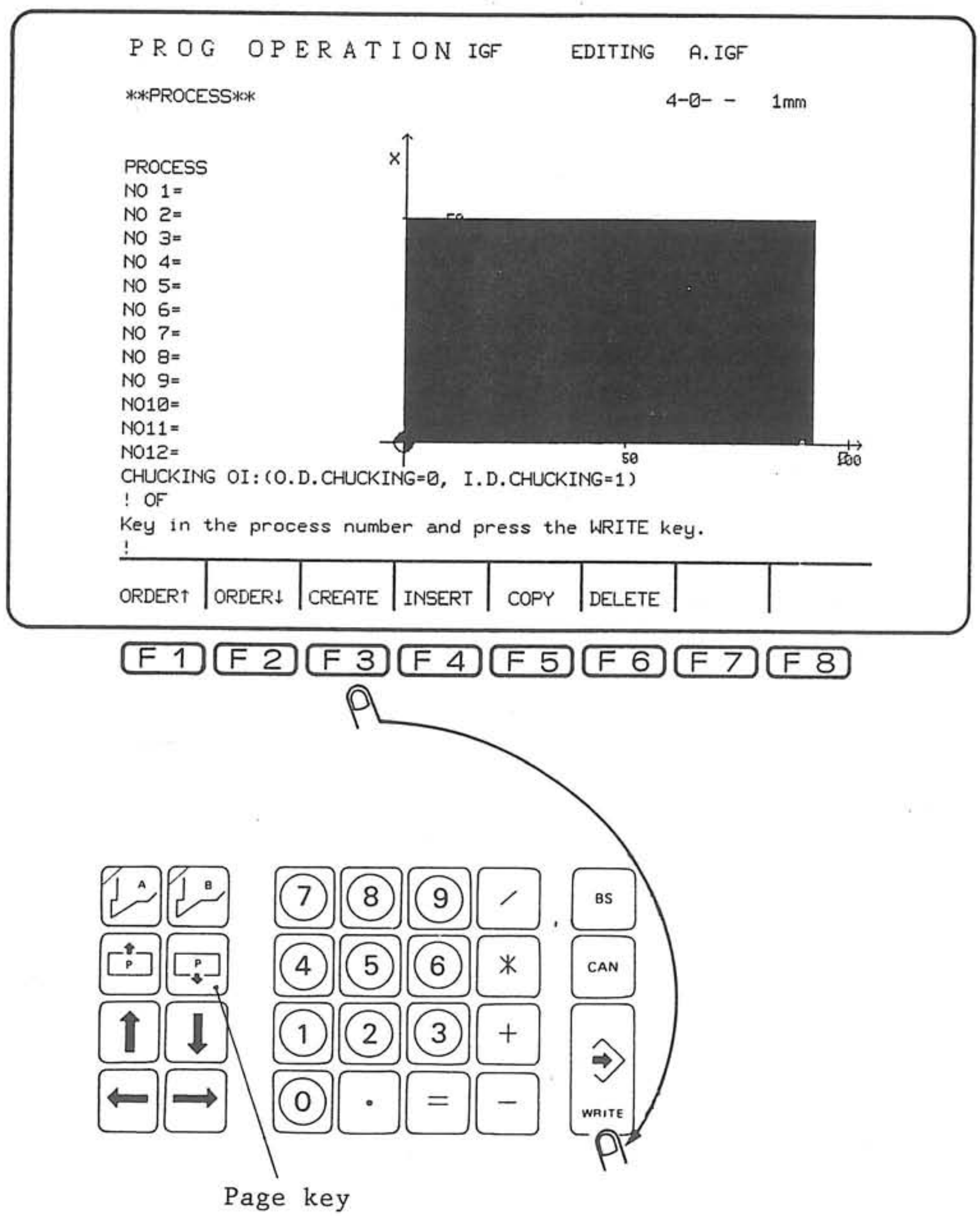
(1) PROCESS Designation (ROUGH O.FACE ↓)

The processes having been registered are displayed in the order of registration.

The first page of the **PROCESS** displays the first 12 sets of the processes and the next 12 sets of the processes (NO 13 through NO 24) can be displayed by pressing the page key [P↓].

Operation Procedure:

a) Press the function key [F3] (CREATE) since the process is newly designated. Then, press the WRITE key to advance to the process (kind) process.



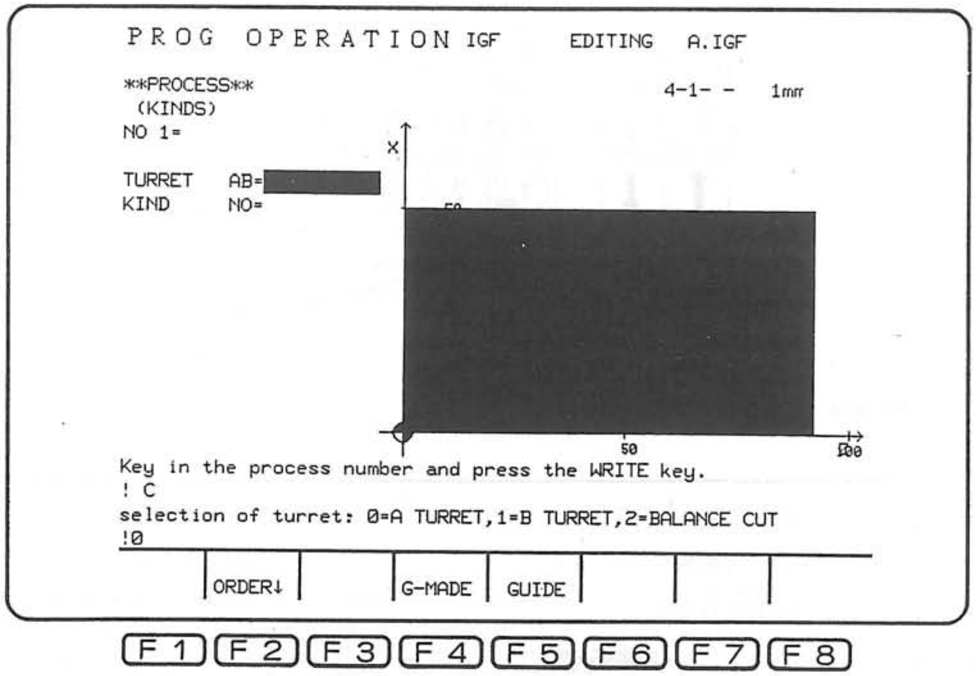
For other function keys [INSERT], [COPY] and [DELETE], refer to the Application Manual (Publication No. 2476-E).

In this step, designation of a process number is allowed. When the designation of a process number is omitted, the process is assigned with the unused and the smallest process number.

>GM (<process-number>) [WRITE]

omissible

(2) PROCESS (KINDS) Designation (ROUGH O.FACE ←)

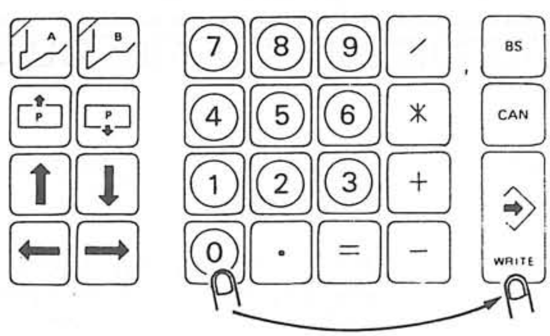


As the NO 1 process, select the ROUGH process.

a) For the 2-saddle or 2-turret model, select the turret to use. ("Ø" for A-turret and "1" for B-turret)

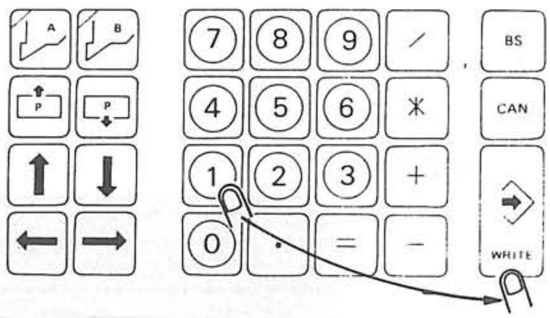
In this example, A-turret is selected.

Press the numerical key "Ø" and the WRITE key.

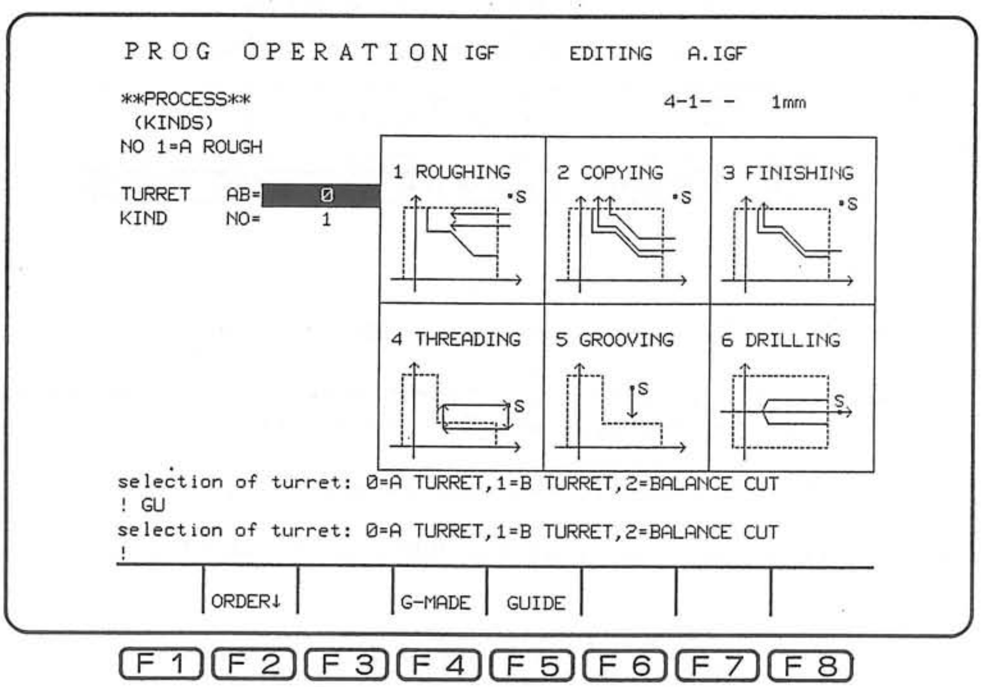


b) Select "1 ROUGHING" as the process kind.

Press the numerical key "1" and the WRITE key.



Other function keys:



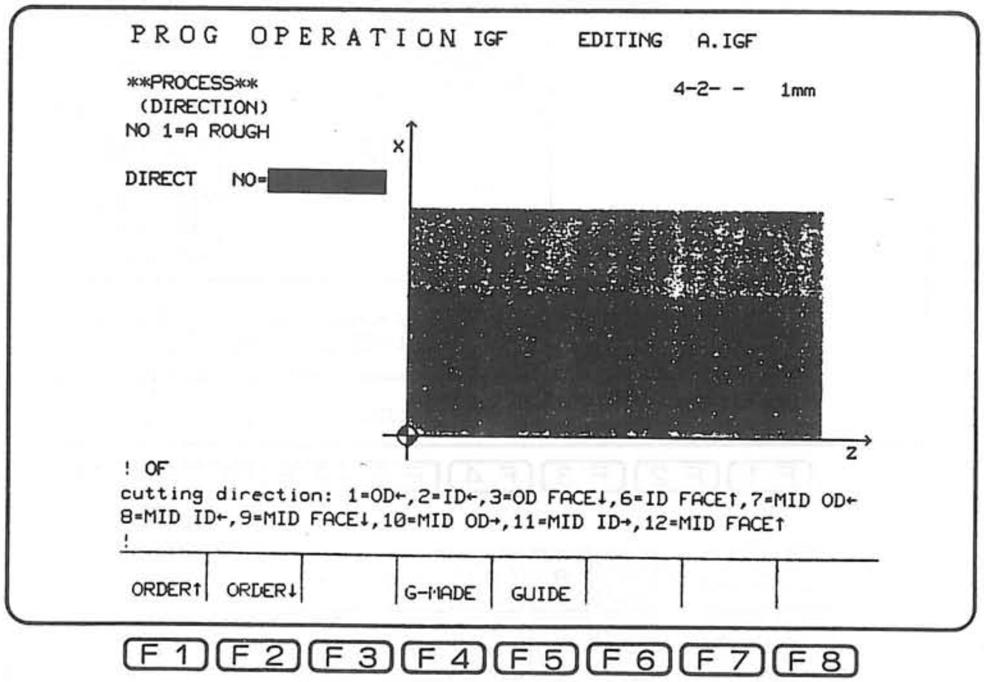
[F4] (G-MADE) Processes having been defined are all displayed.

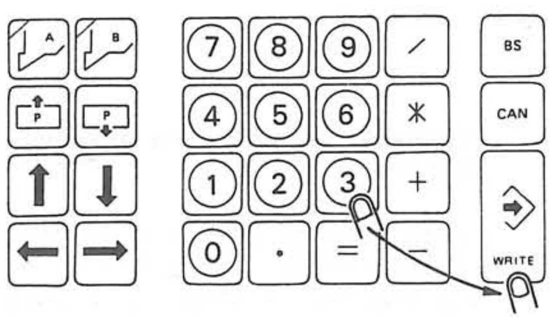
[F5] (GUIDE) Guide drawing for process kind selection is displayed. Process kind "7 RECESS" is displayed when the key is pressed.

(3) PROCESS (DIRECTION) Designation (ROUGH O.FACE ↓)

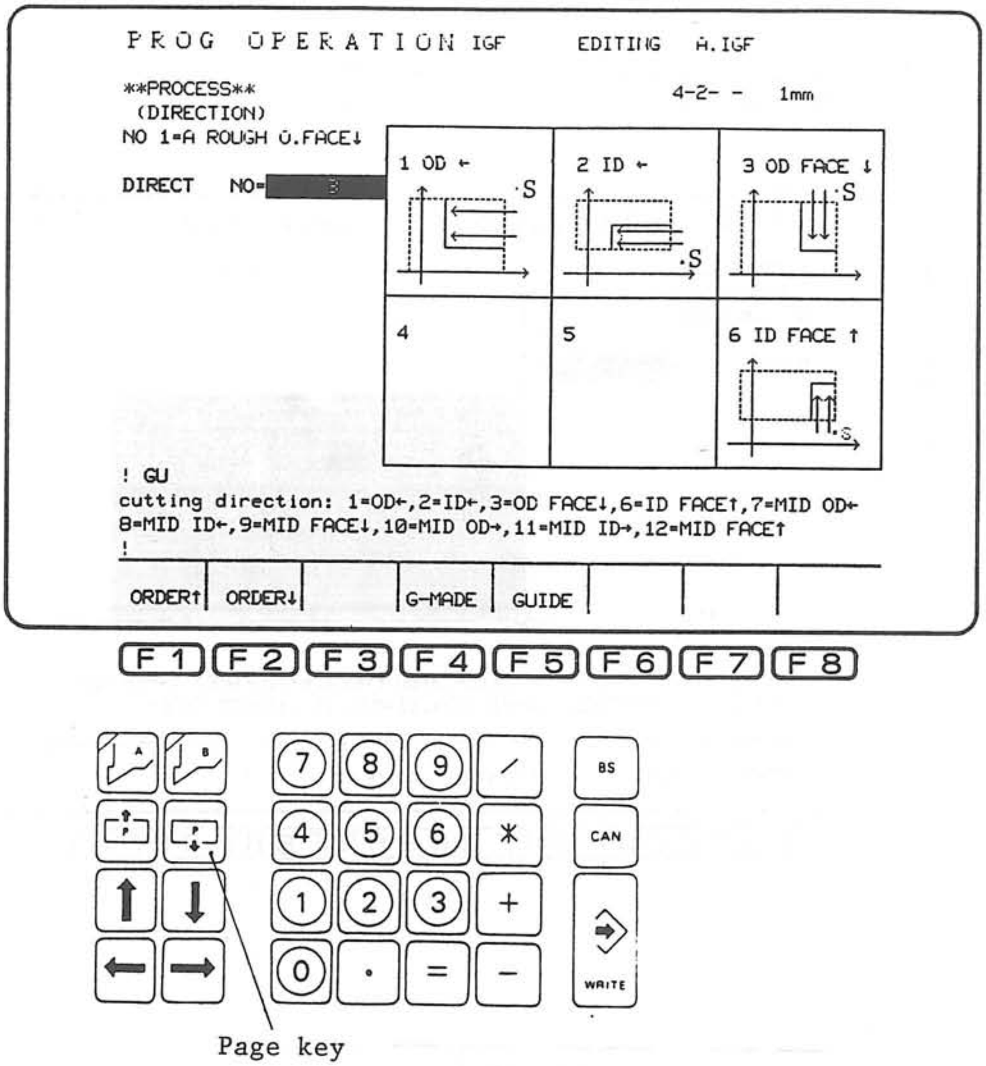
As a direction of cut, select OD FACE ↓.

a) The direction of cut is "3=OD FACE ↓". Enter "3" by pressing the numerical key "3" and the WRITE key.





Other function keys:



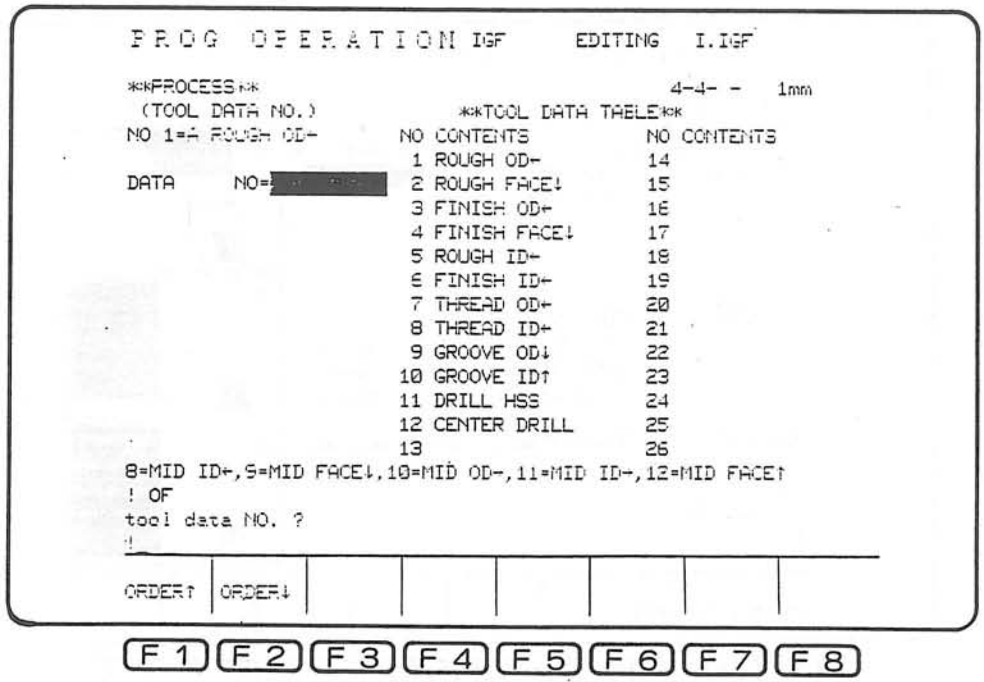
[F4] (G-MADE) Processes having been defined are all displayed.

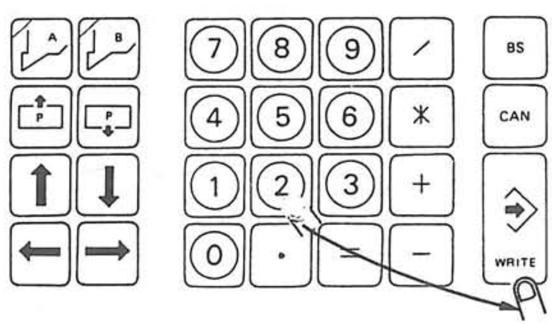
[F5] (GUIDE) Guide drawing for process direction selection is displayed. Two guide drawings are provided and display of them are changed by pressing the page keys.

(4) PROCESS (TOOL DATA NO) Designation (ROUGH O.FACE ↓)

Select the data number of the tool data to be used.

a) The tool to be used is "2=ROUGH FACE ↓ ". Enter "2" by pressing the numerical key "2" and the WRITE key.

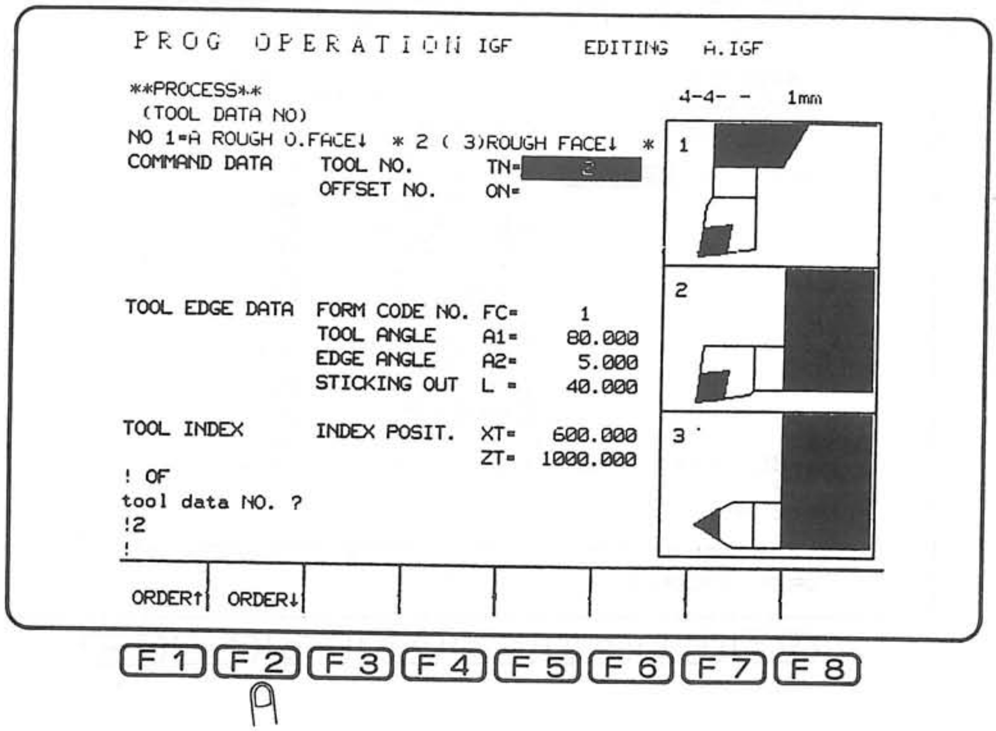




Check the entered data.

Note that the selection of the tool inconsistent with the selected process kind name causes an error.

b) If the tool desired is not found in the tool data table, enter "Ø" at parameter DATA. On the tool data confirmation screen, default values are set to the parameters other than TOOL NO. and OFFSET NO. Set the data only for those parameters for which default values are not set or modification is required. If DATA NO. Ø is set in step a), data must be set for all parameters.



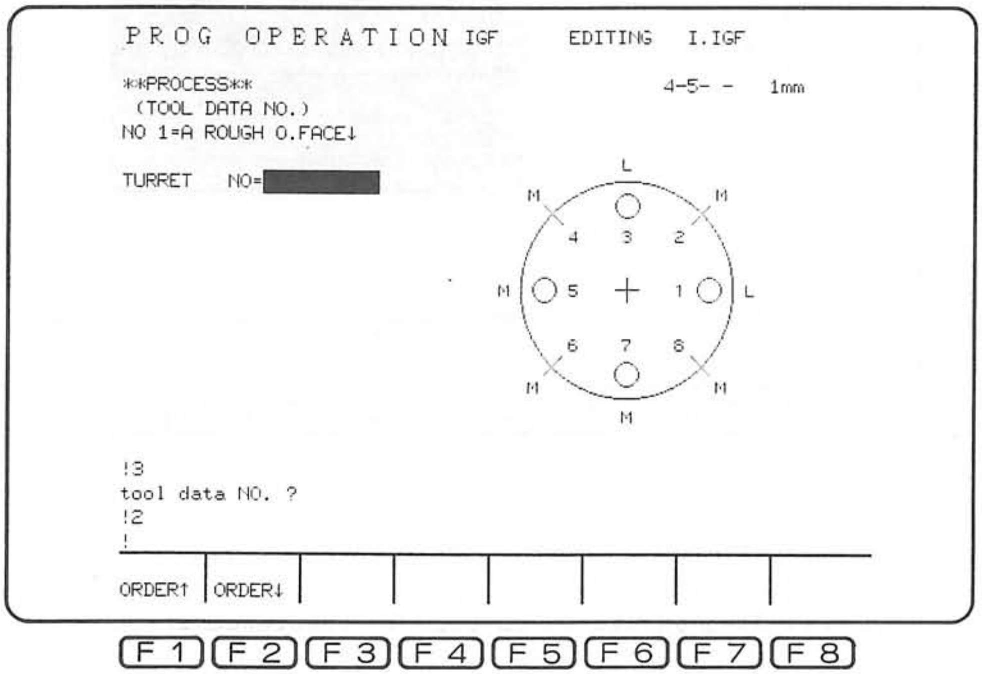
c) Press the function key [F2] (ORDER ↓) to advance to the next process, PROCESS (CONDITION).

Special Tool Designation

For LR15M-ATC, it is necessary to set the turret number on which the selected tool is set or to be set with ATC, after the designation of a tool data number and before setting the tool data.

For tools used for turning operations, designate a turret number which is appended with L and for those used for milling operations, designate a turret number which is appended with M.

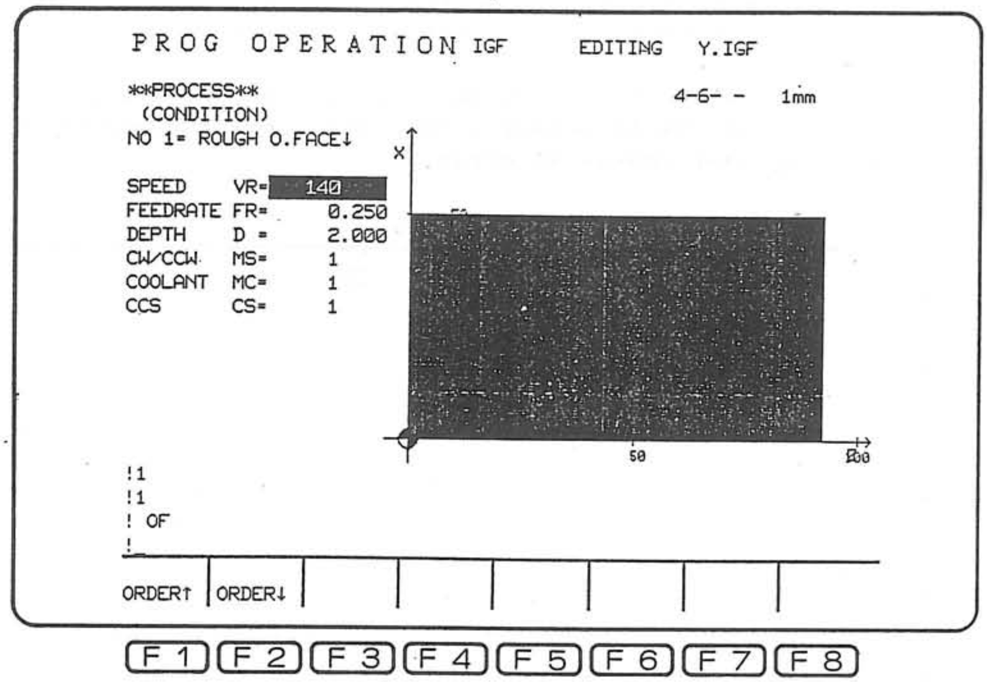
An attempt to select M indicated turret number for designating a turning tool and to select L indicated turret number for designating a milling tool causes an error.



(5) PROCESS (CONDITION) Designation (ROUGH O.FACE ↓)

Cutting conditions automatically determined based on the data entered from the preceding processes are displayed.

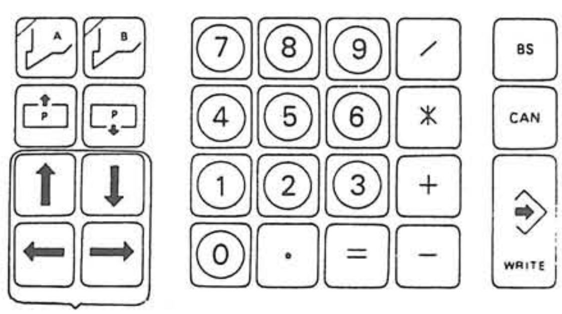
Check the cutting conditions.



If the data displayed is to be edited, locate the cursor on it and key in the desired data. For parameters CW/CCW and COOLANT, required data should be entered in reference to the menu, which is displayed when the cursor is located on these parameters.

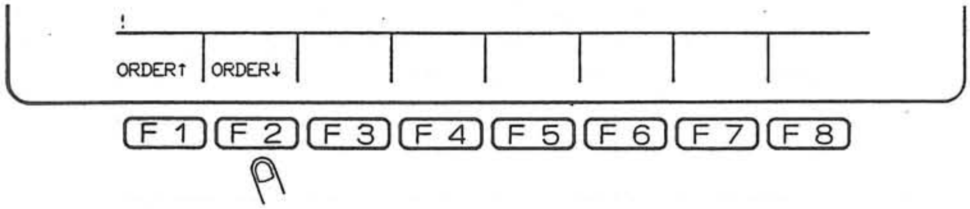
In the case of constant cutting speed,

Constant cutting speed control CS = Ø: OFF
1: ON

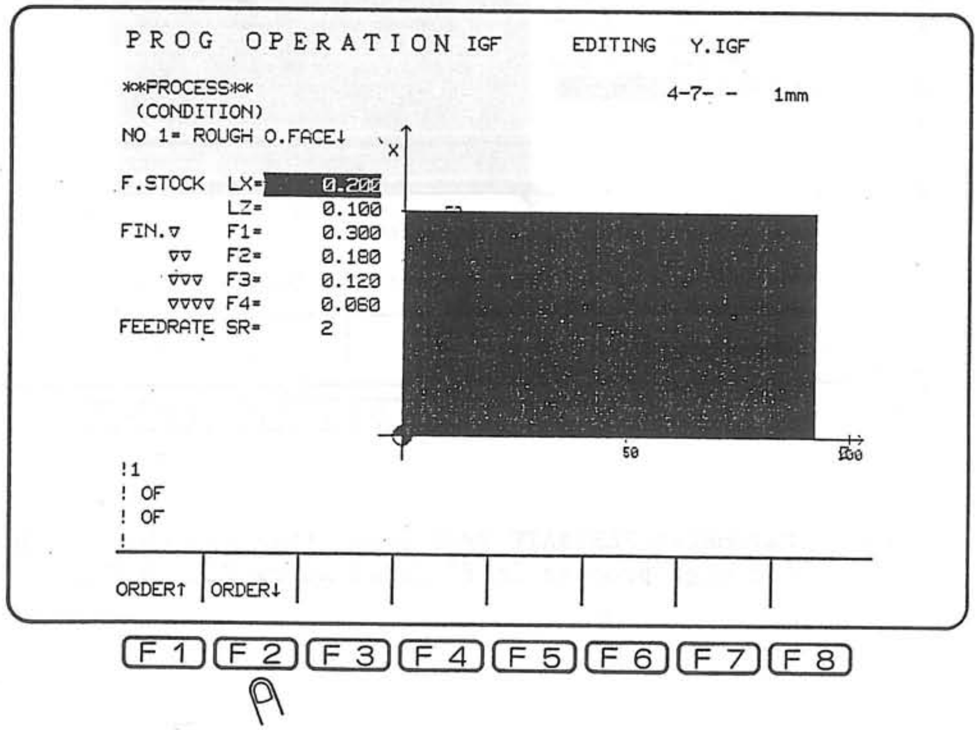


Cursor keys

a) After confirming the correctness of the cutting conditions displayed, press the function key [F2] (ORDER ↓).

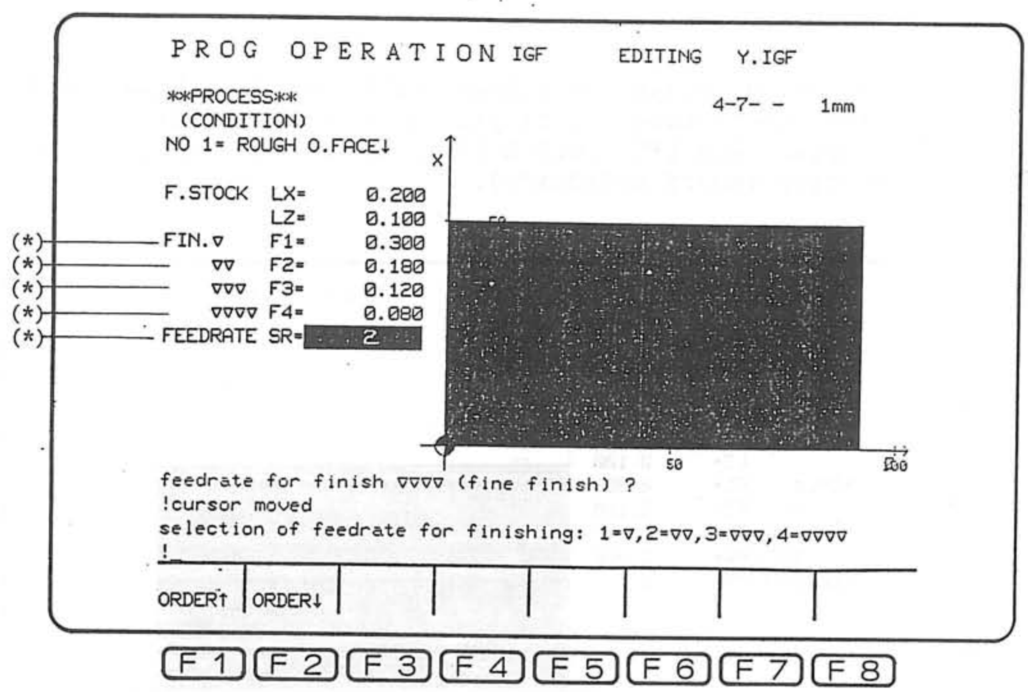


b) The CRT displays the automatically set finishing conditions. After confirming the displayed cutting conditions, press the function key [F2] (ORDER ↓) to advance to the next process, PROCESS (SHAPE DESIGNATE).



If displayed cutting conditions are to be edited:

If any of cutting conditions displayed is to be edited for optimizing the turning operation, locate the cursor on the data to be edited and then key in the desired data directly. For the parameters (*) indicated below, menus are displayed when the cursor is placed on them.

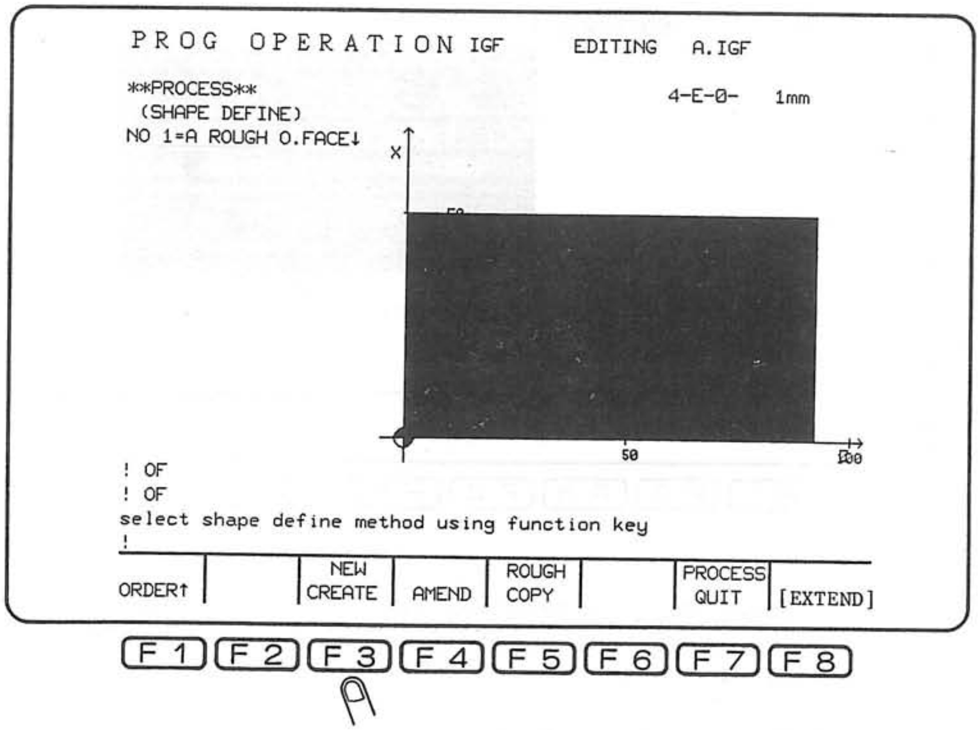


Note: Parameter FEEDRATE SR=2 means that the shape to be defined in the next process is finished at ▽▽ F2 = Ø.18.

(6) PROCESS (SHAPE DEFINE) Designation (ROUGH O.FACE ↓)

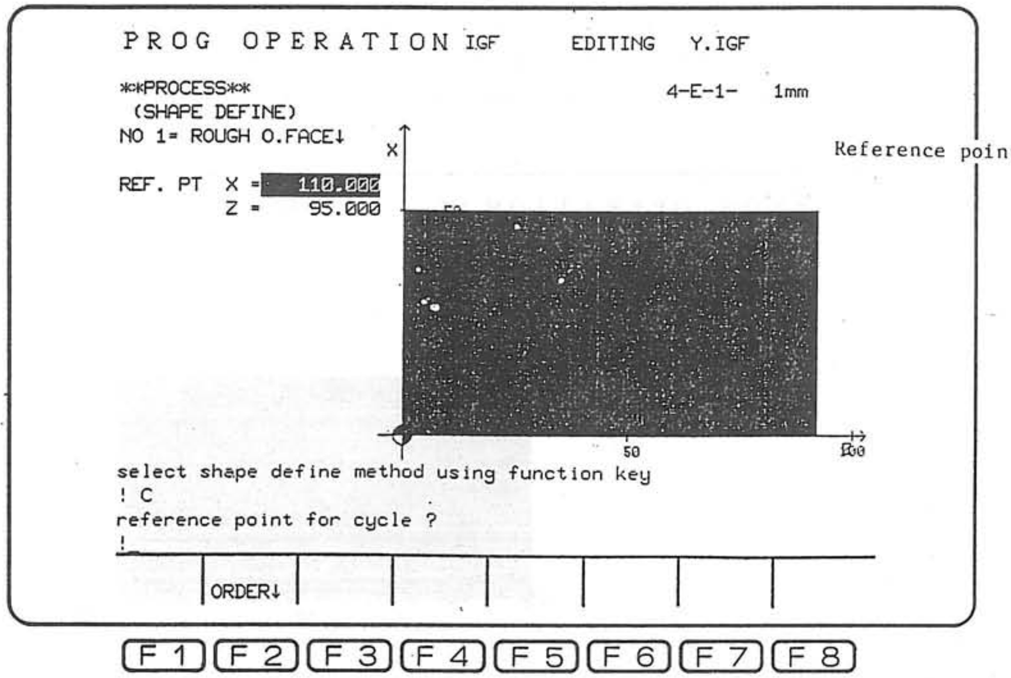
Select whether the shape to be input is a new shape and all related data is entered or it is defined by editing the data stored.

a) In this example, the shape for ROUGH O.FACE ↓ PROCESS is newly created, press the function key [F3] (NEW CREATE). This prompts the operator to set the cycle reference point (REF. PT).



For other function keys, refer to the Application Manual (Publication No. 2476-E).

The cycle reference point (REF. PT) is automatically determined from the blank shape and clearance amount in X and Z directions. Check the data displayed.

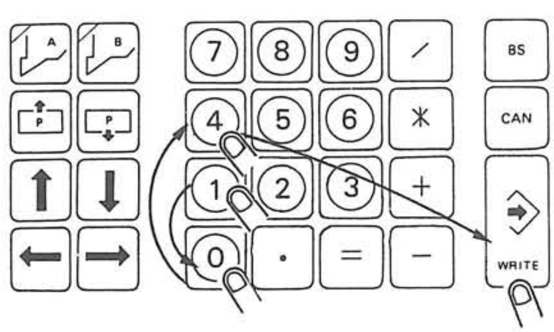


The REF. PT is determined assuming the clearance of ϕ1Ø mm in the radial direction and 3 mm in the longitudinal direction. Change the data meeting the intended cutting.

In this example, the clearance should be ϕ4 mm (in the radial direction) and Ø mm in the longitudinal direction.

REF. PT X = 1Ø4
Z = 92

b) Enter "1Ø4" for X by pressing the numerical keys "1", "Ø" and "4" in this order and then pressing the WRITE key.



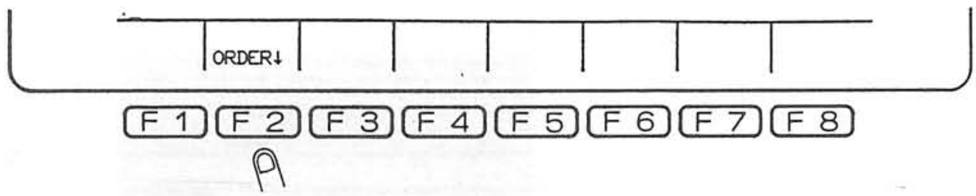
Entry of "1Ø4"

After the entry of the data, the cursor automatically advances to the next data entry position.

c) Enter Z=92 in the same manner.

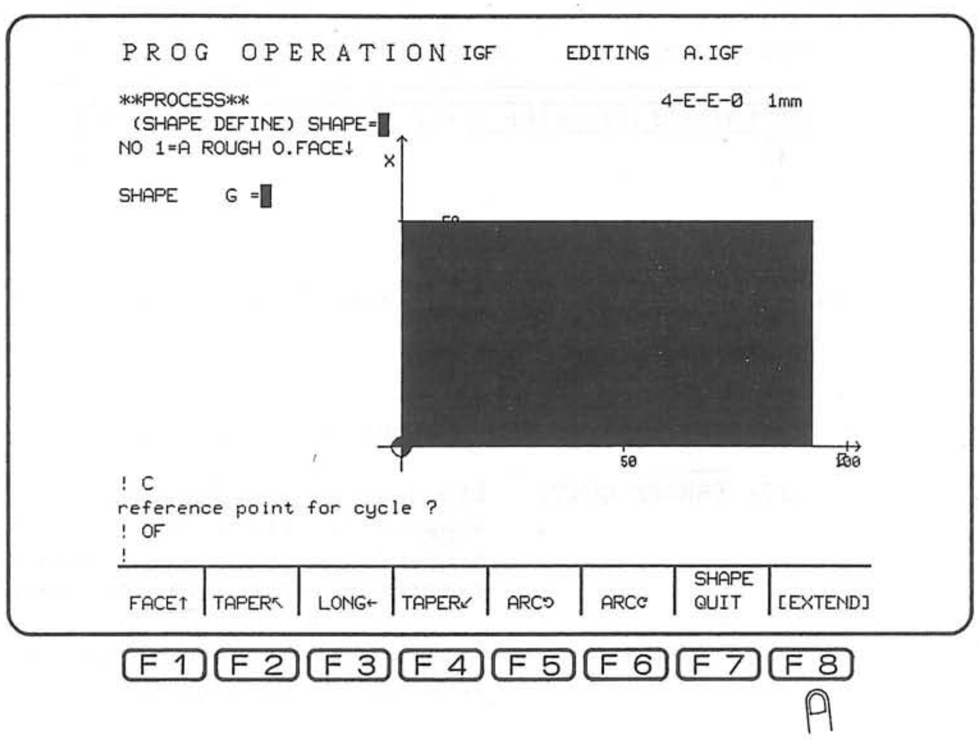
Note: Clearance amounts used for determining the reference point can be changed by parameters.

d) Press the function key [F2] (ORDER↓) to advance to the SHAPE element entry process.



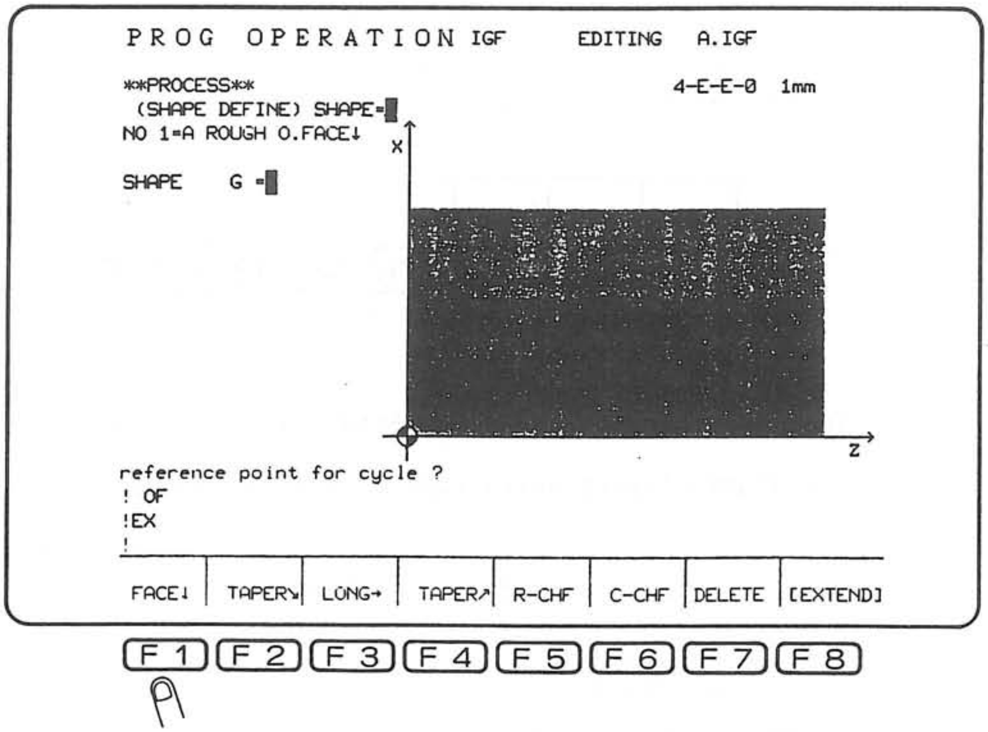
e) The shape element can be entered using the function keys.

The SHAPE element entry page is indicated below:



Press the function key [F8] (EXTEND).

f) The function key names will be changed.

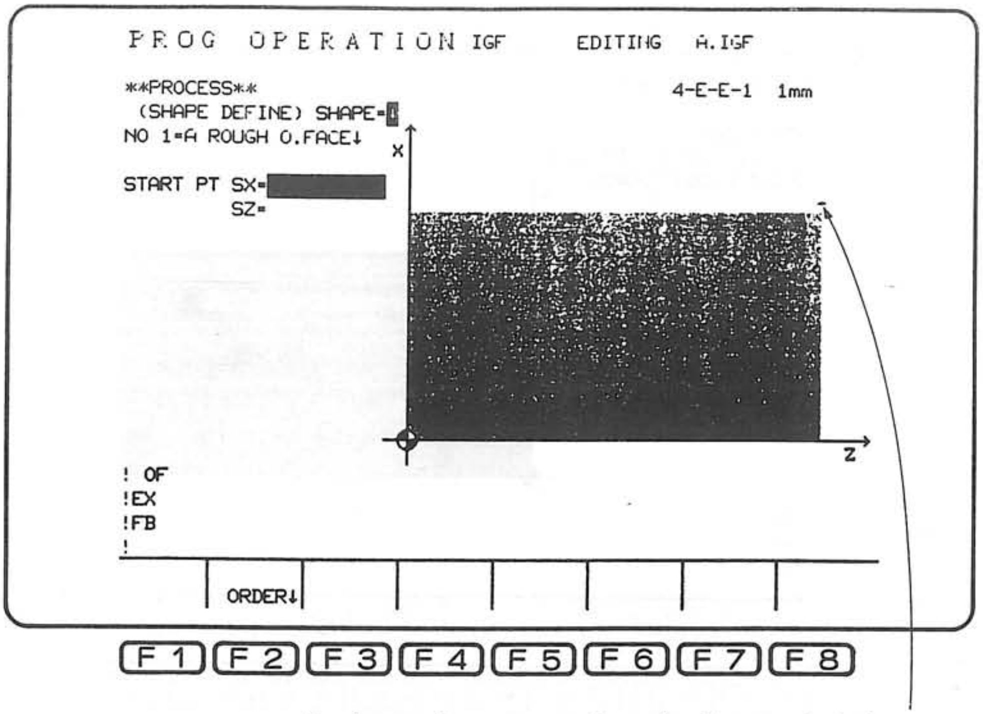


Press the function key [F1] (FACE ↓) to begin the start point (START PT) data entry.

Note: Function keys used for other than shape element designation.

[F7] (SHAPE QUIT)	Press this key after the completion of shape definition. When this key is pressed, the display is returned to the first page of the PROCESS (SHAPE DEFINE).
[F7] (DELETE)	This key deletes the shape element located by the cursor.

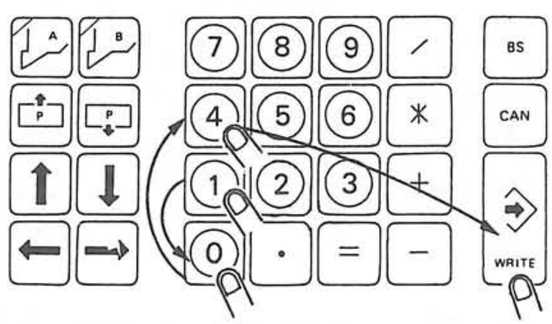
g) Enter the START PT data. This entry is required only at the entry of the first shape element.



Cycle reference point designated flickers

Assume the start point of SX=1Ø4, SZ=9Ø.

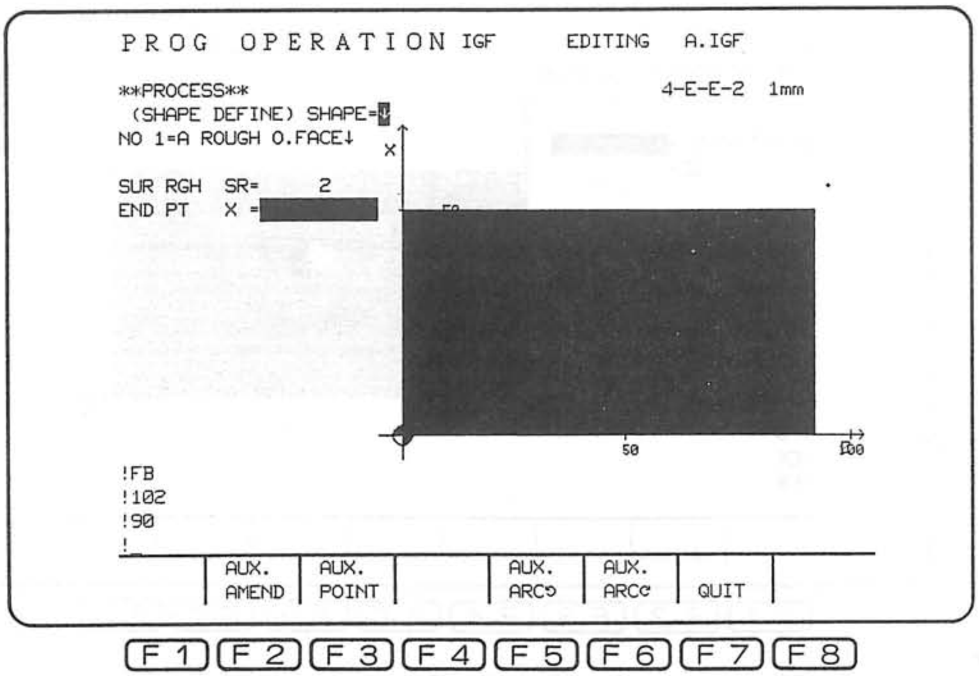
After making sure that the cursor is on the parameter START PT SX=, key in "1", "Ø", "4" and press the WRITE key.



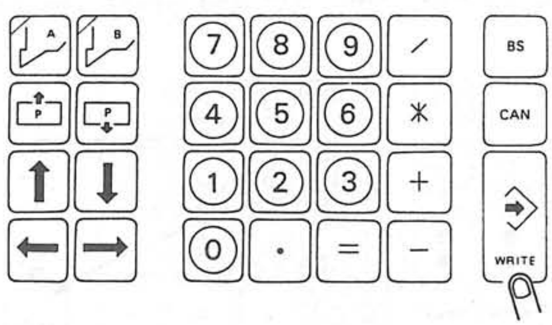
Entry of "1Ø4"

h) Enter SZ=9Ø in the same manner.

The step to be followed is the entry of shape element data corresponding to the shape element selected. It is not necessary to enter unknown data or values not indicated on a part drawing.



i) Make sure that parameter SUR RGH SR= has default of "2"* and that the cursor is at parameter END PT X.

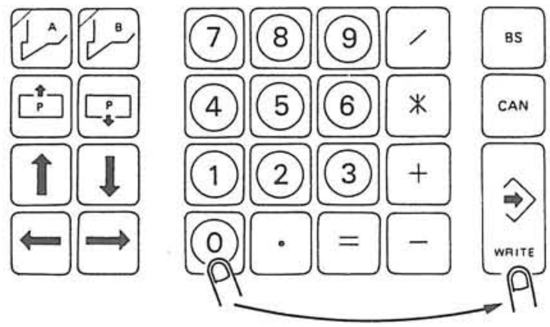


* Default "2" is provided here since "SR=2" is set in the PROCESS (CONDITION) page.

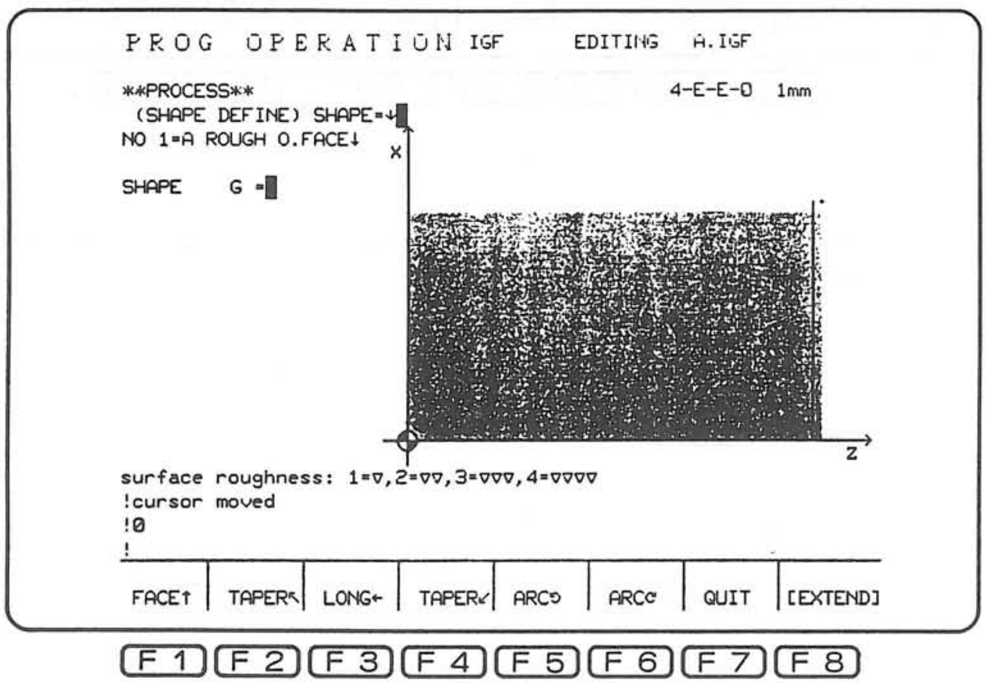
If surface roughness is to be changed, return the cursor to parameter SUR RGH SR.

j) Enter the end point (END PT X) data.

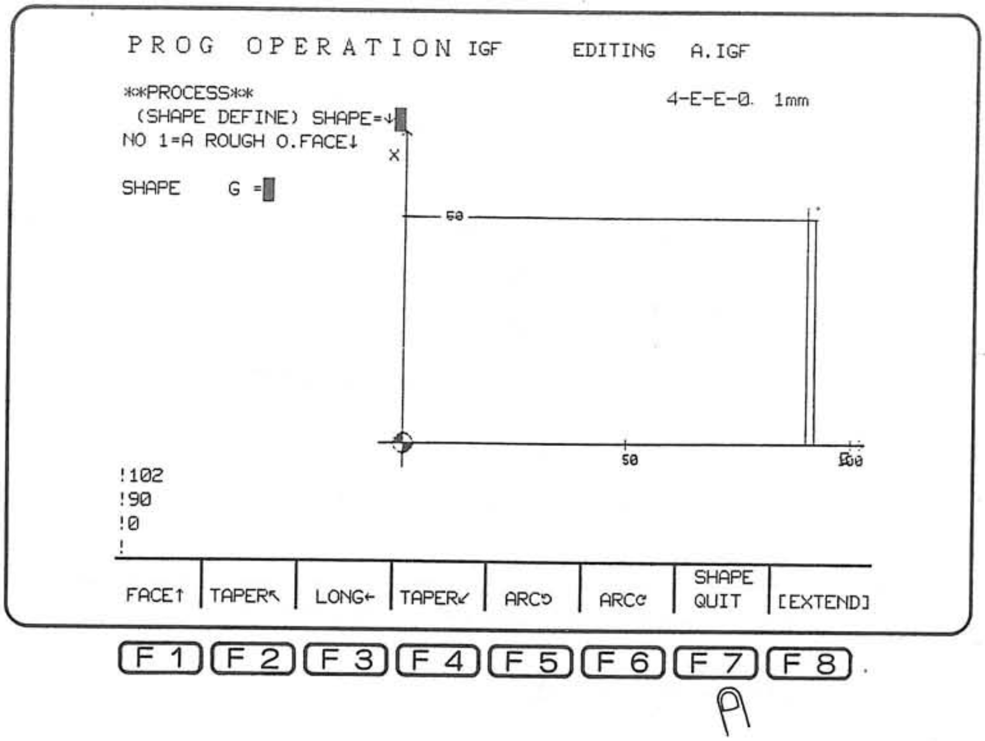
In this example, set "Ø" by pressing the numerical key "Ø" and the WRITE key.



The display will be changed as indicated below:

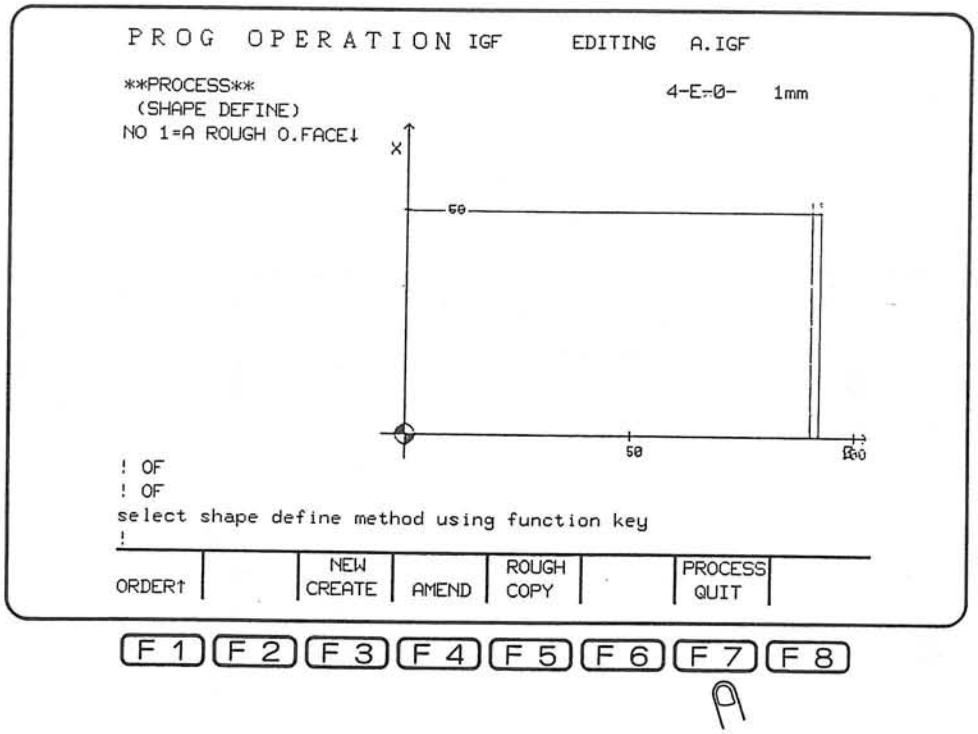


k) Press the function key [F7] (SHAPE QUIT).

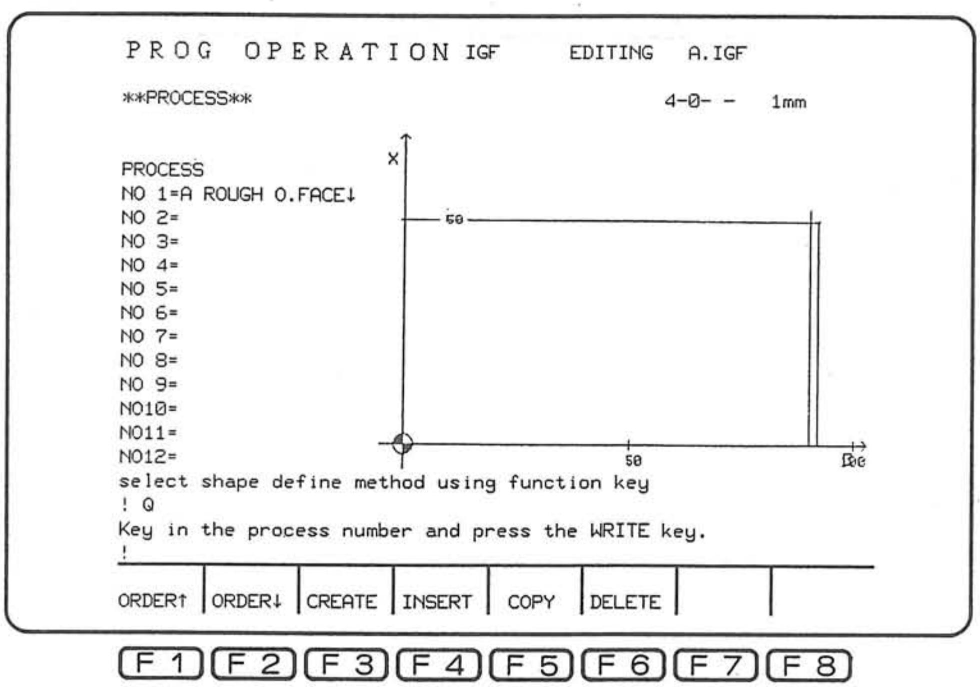


This returns the display to the first step of the PROCESS (SHAPE DEFINE) process.

1) Make sure that the display is the first page of the PROCESS (SHAPE DEFINE) process. Press the function key [F7] (PROCESS QUIT).

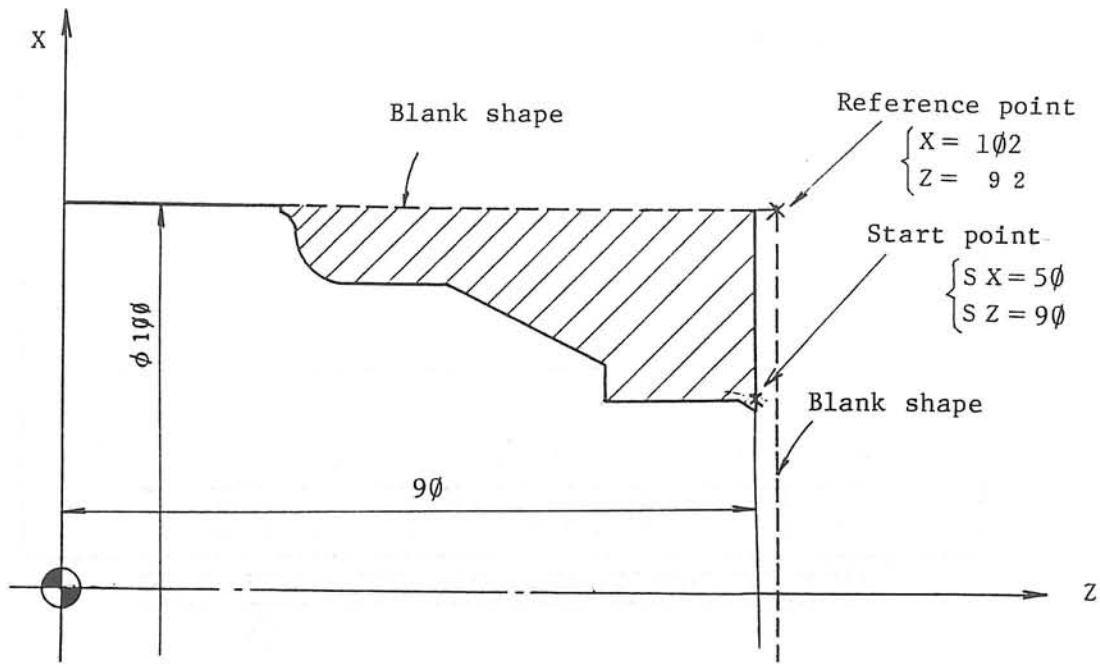


The page should be the PROCESS page. This completes the ROUGH O.FACE ↓ process registration.



7-6-2. Registering Rough OD Process

The ROUGH OD ← process removes the portion indicated by hatching lines illustrated in Fig. 7-6-2.

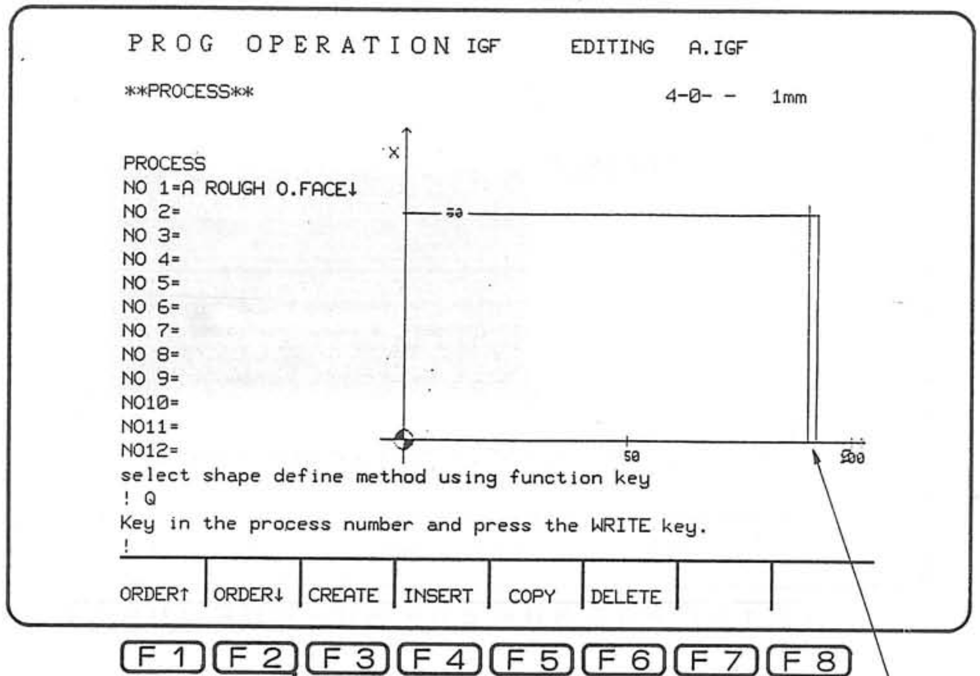


PROCESS NO. 2 ROUGH OD ←

Fig. 7-6-2 Rough OD Turning Process

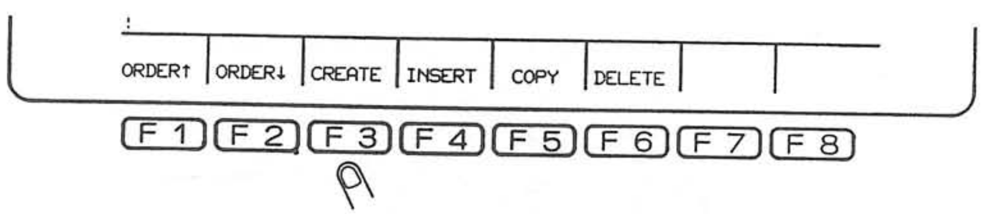
(1) PROCESS Designation (ROUGH OD←)

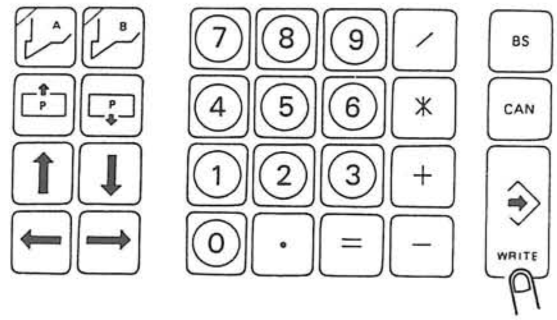
Enter the rough OD← turning process data next.



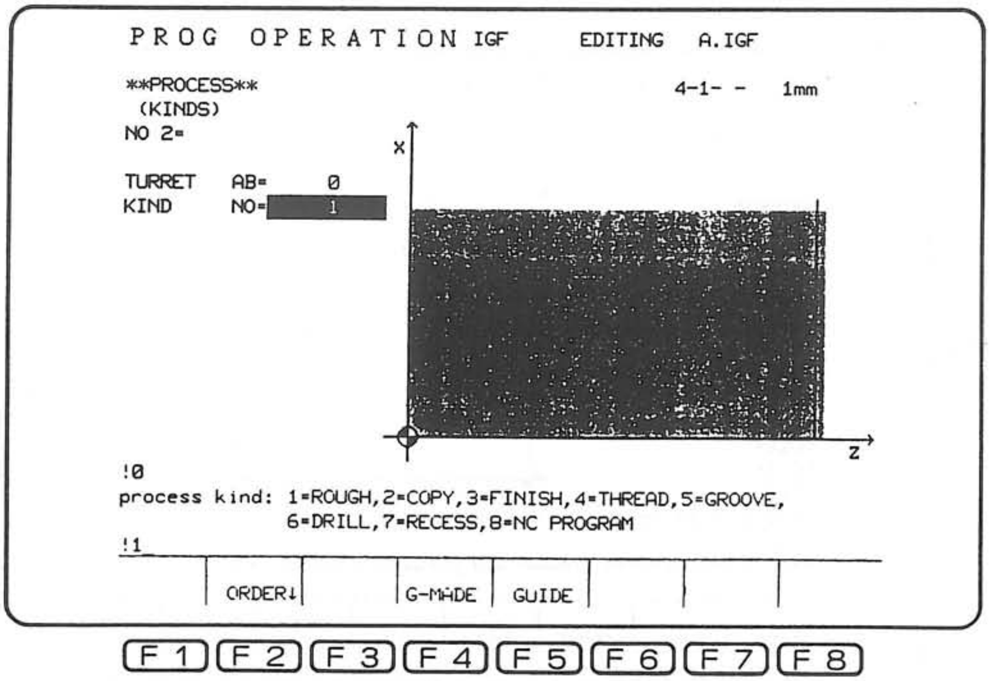
The shape defined in the ROUGH 0. FACE ↓ is displayed.

a) Press the function key [F3] (CREATE) and the WRITE key.





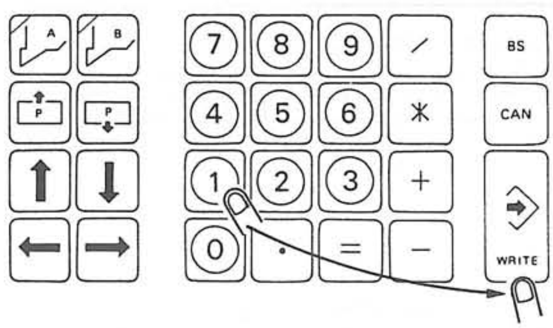
(2) PROCESS (KINDS) Designation (ROUGH OD ←)



As the NO 2 process, select the ROUGH process.

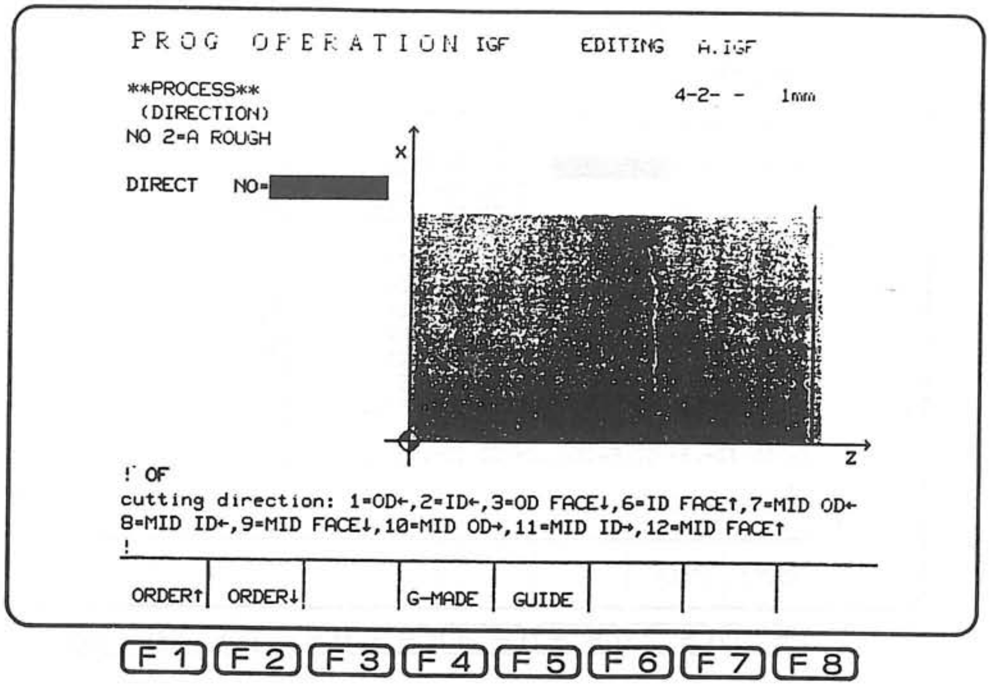
a) Select "1 ROUGH" as the process kind.

Press the numerical key "1" and the WRITE key.

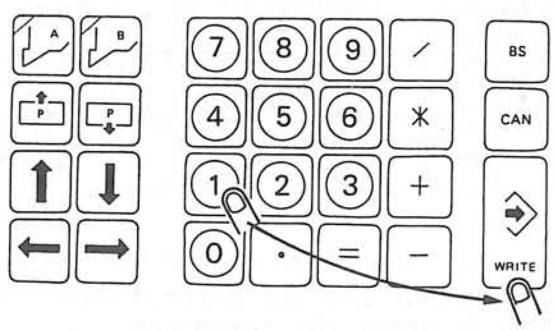


(3) PROCESS (DIRECTION) Designation (ROUGH OD ←)

As a direction of cut, select "OD ←".

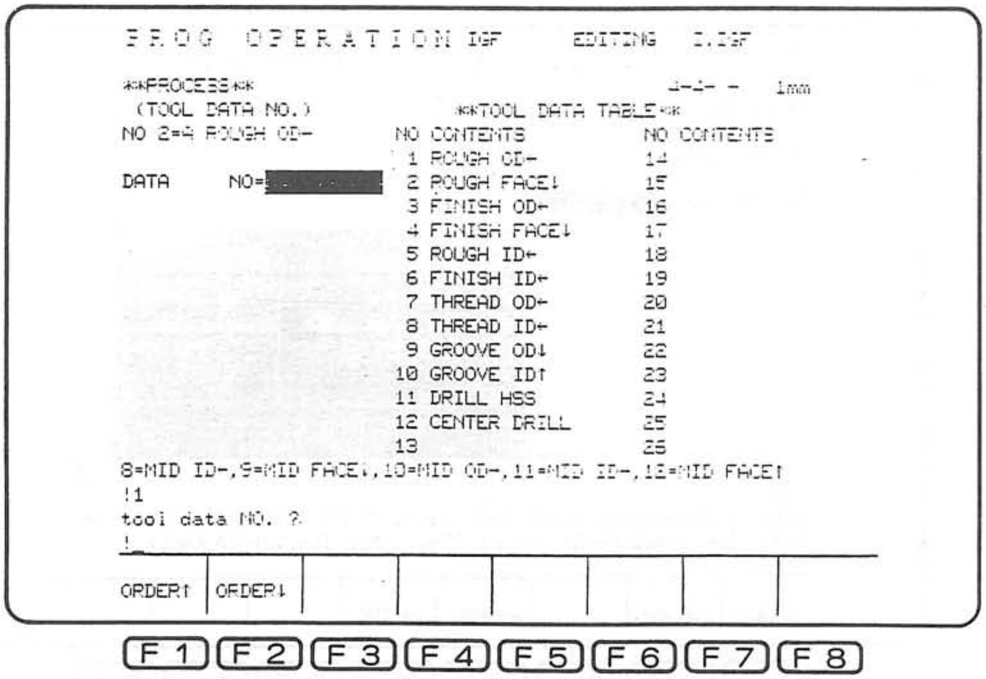


a) The direction of cut is "1=OD ←". Enter "1" by pressing the numerical key "1" and the WRITE key.

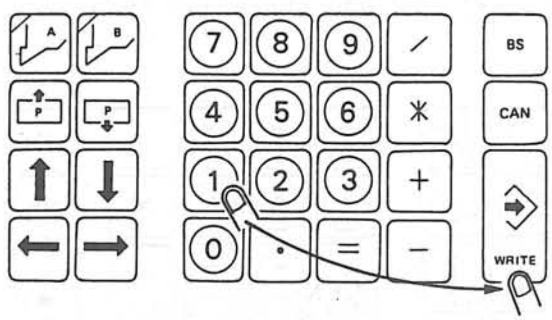


(4) PROCESS (TOOL DATA NO) Designation (ROUGH OD←)

Select the data number of the tool data to be used.

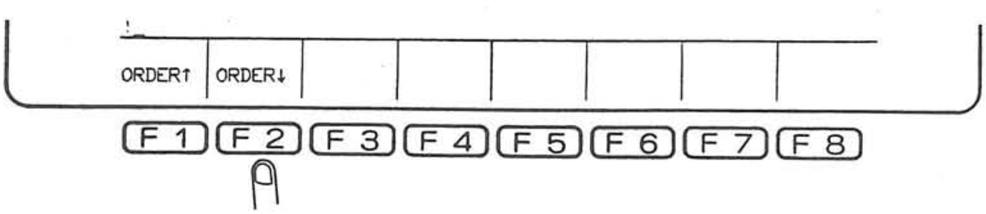


a) The tool to be used is "1 ROUGH OD←". Enter "1" by pressing the numerical key "1" and the WRITE key.

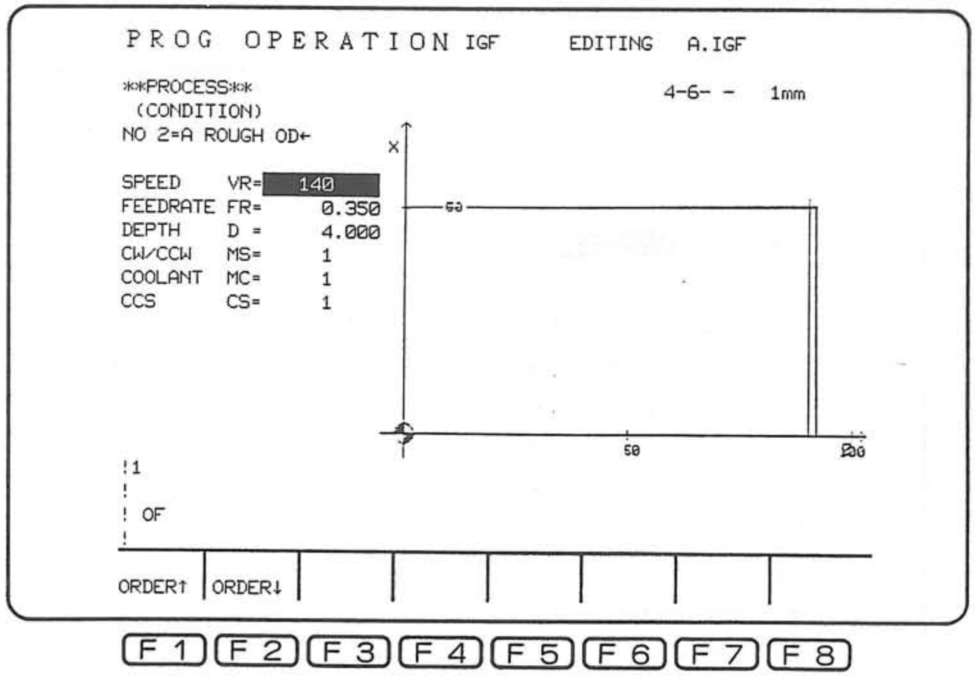


Make sure that the entered DATA NO "2" is displayed. If the data displayed is not the entered data, make corrections on the data after locating the cursor on it.

b) Press the function key [F2] (ORDER↓).



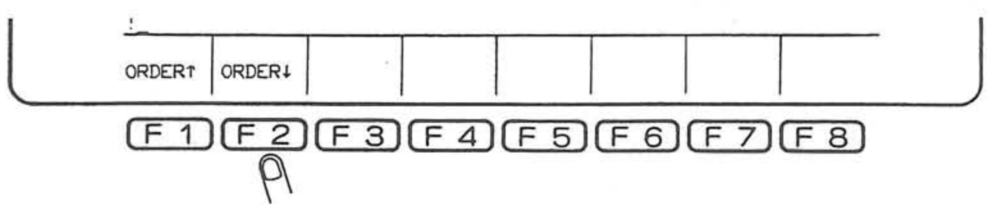
(5) PROCESS (CONDITION) Designation (ROUGH OD ←)



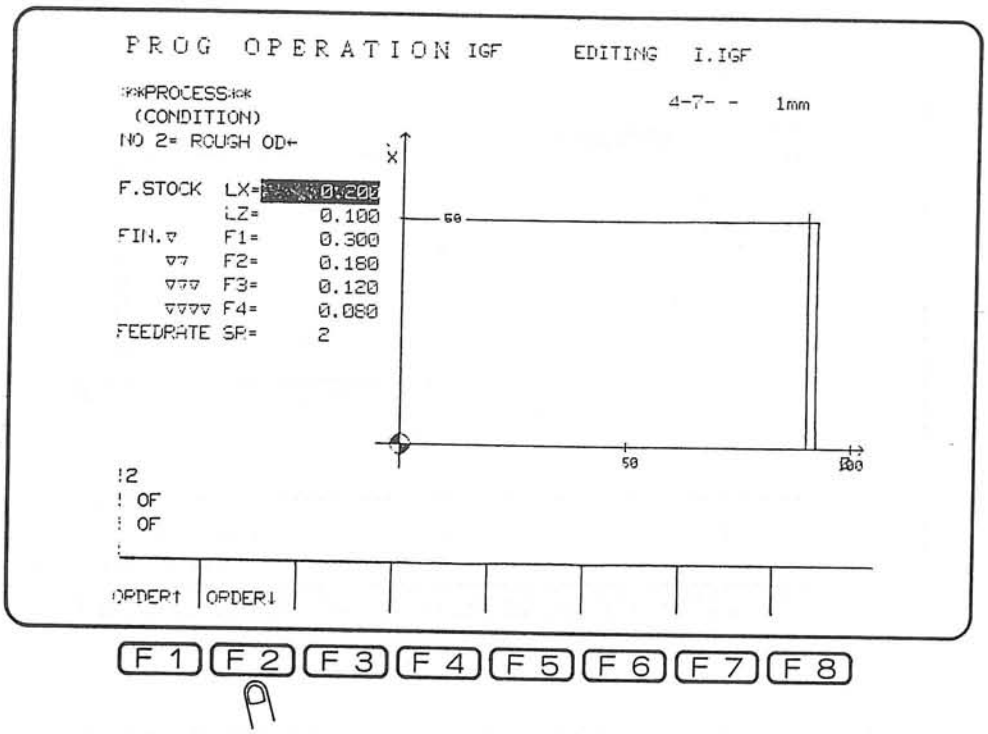
Cutting conditions automatically determined based on the data entered from the preceding processes are displayed.

a) Check the cutting conditions and make corrections on them if necessary.

Press the function key [F2] (ORDER ↓).



b) The CRT displays the automatically set finishing conditions. Check the conditions displayed and make corrections on them if necessary.



Press the function key [F2] (ORDER ↓) to advance to the next process PROCESS (SHAPE DEFINE).

(6) PROCESS (SHAPE DEFINE) Designation (ROUGH OD ←)

The shape to be defined for ROUGH OD ← is illustrate below.

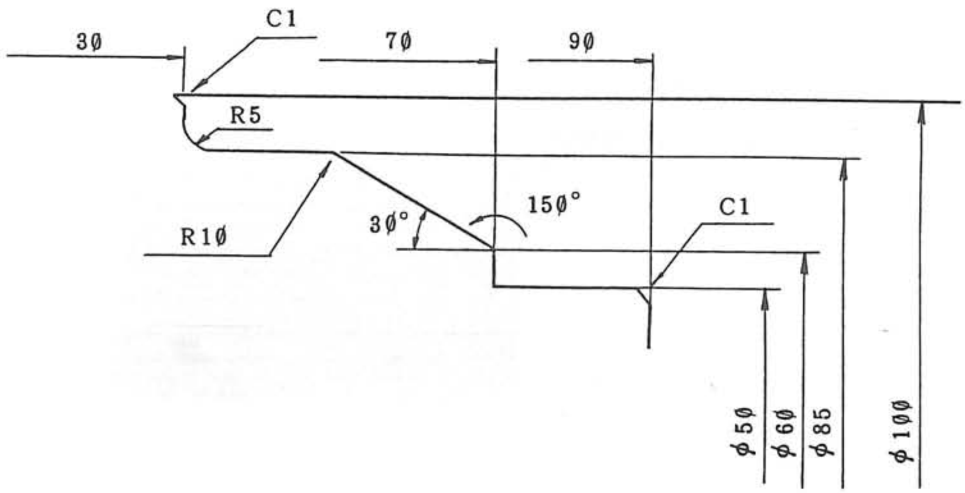
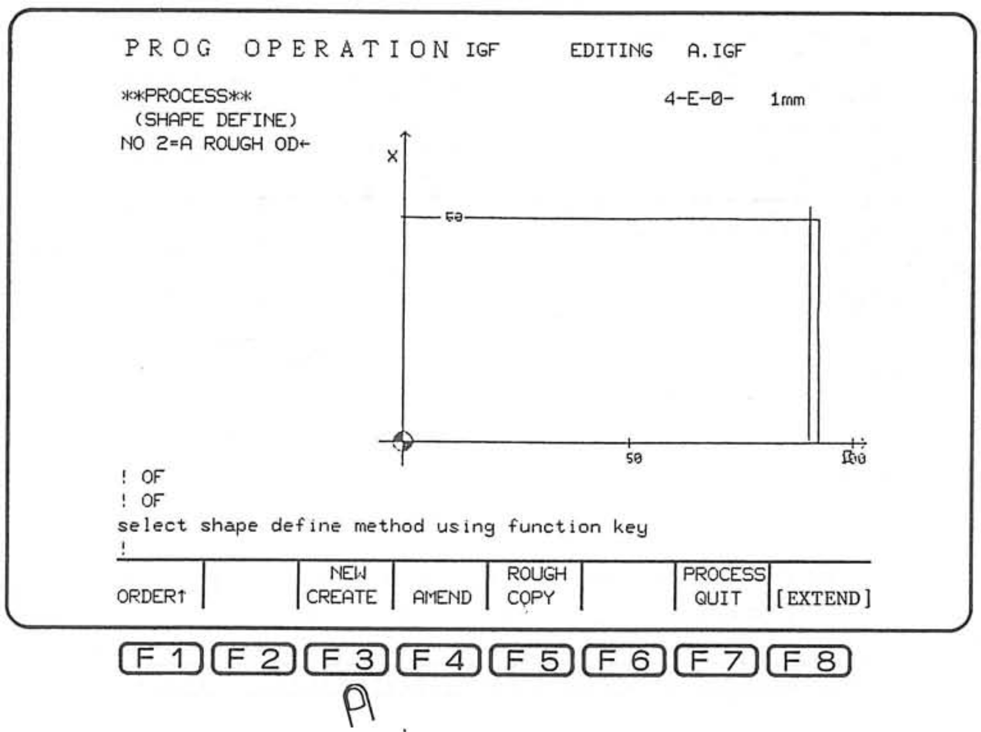


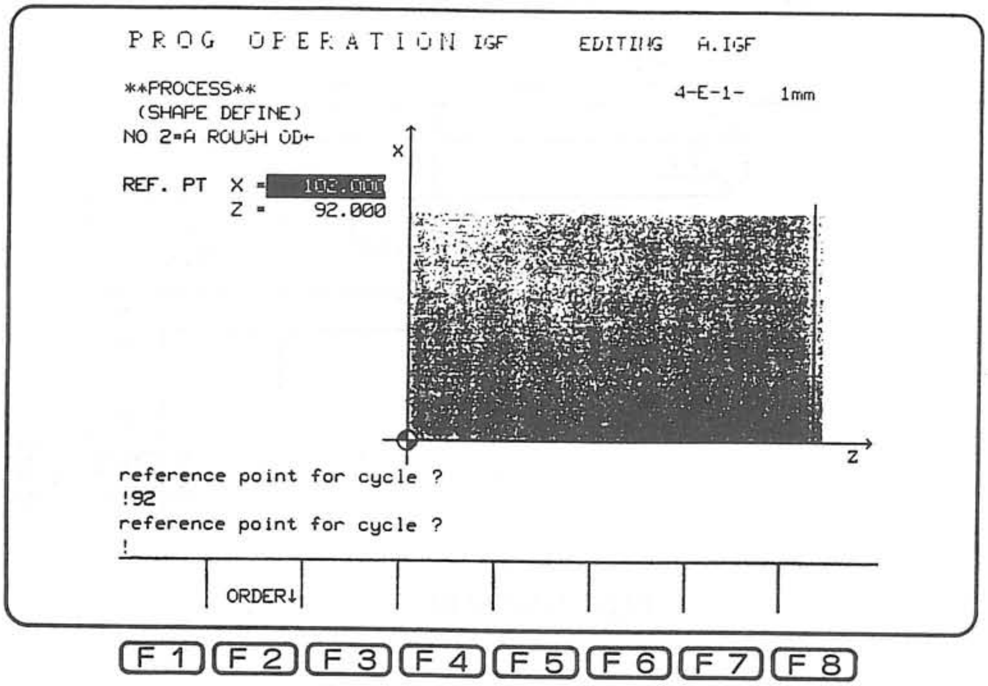
Fig. 7-6-2-(1)

The procedure to define the shape is explained below.

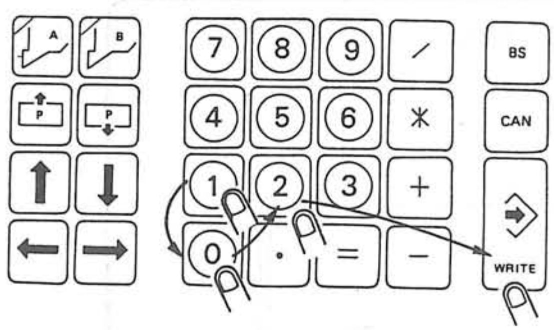
a) Press the function key [F3] (NEW CREATE).



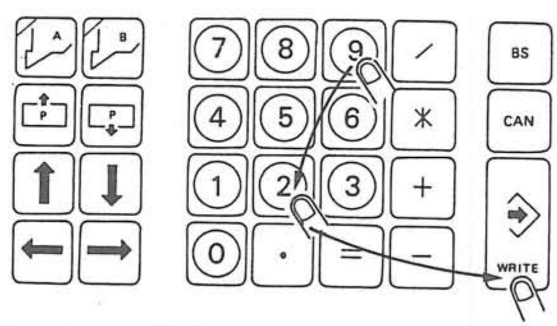
b) Define the reference point (REF. PT).



Enter "1Ø2" for "X" and "92" for "Z".

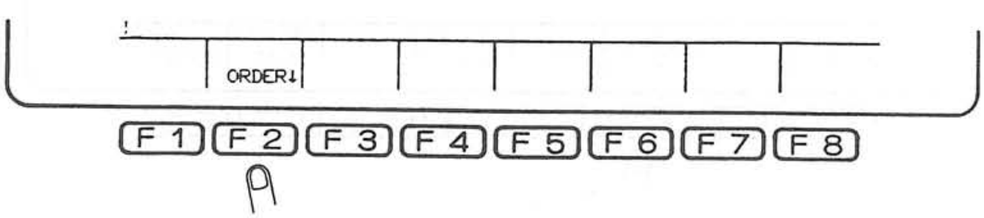


Entry of "1Ø2"

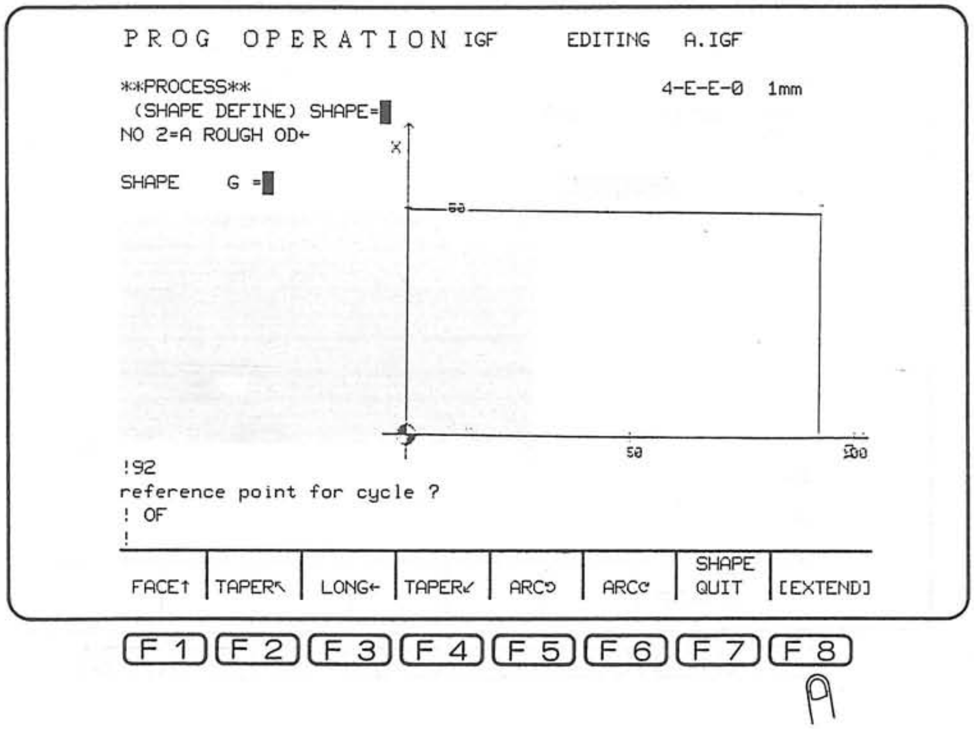


Entry of "92"

c) Press the function key [F2] (ORDER ↓) to advance to the SHAPE element entry process.

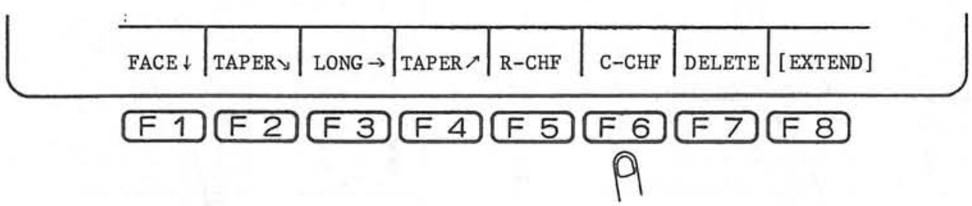


d) The display should be as indicated below.

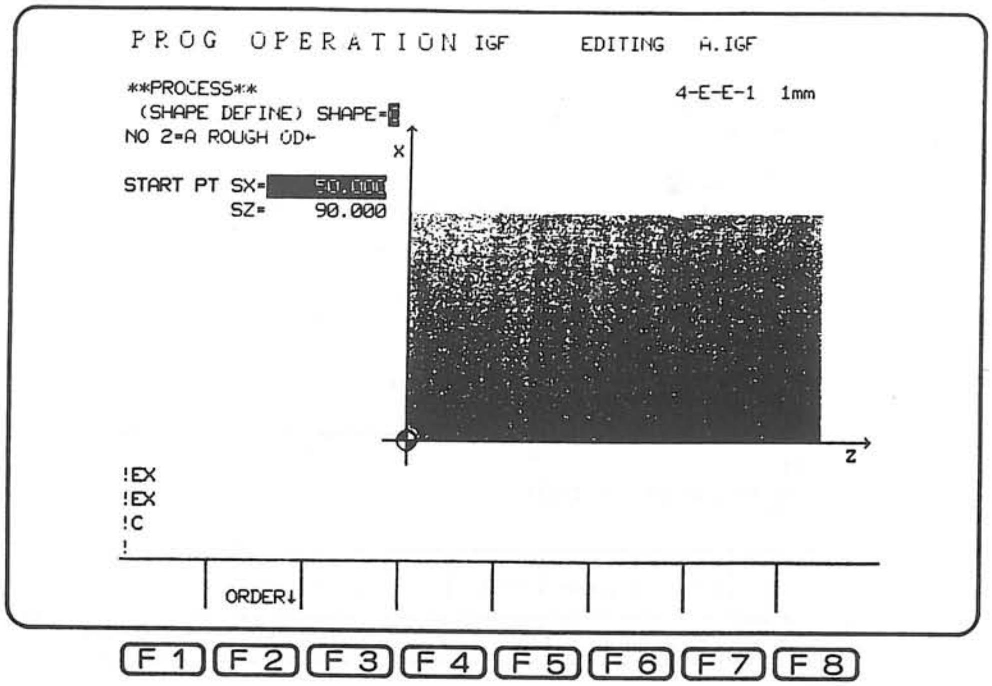


Press the function key [F8] (EXTEND).

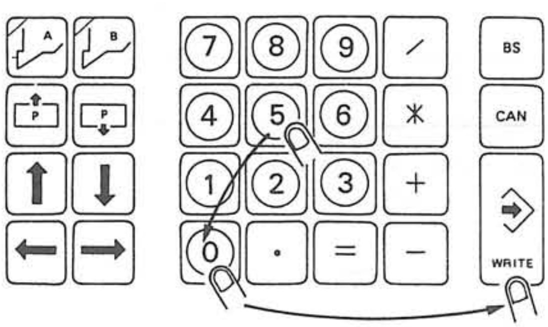
e) Press the function key [F6] (C-CHF).



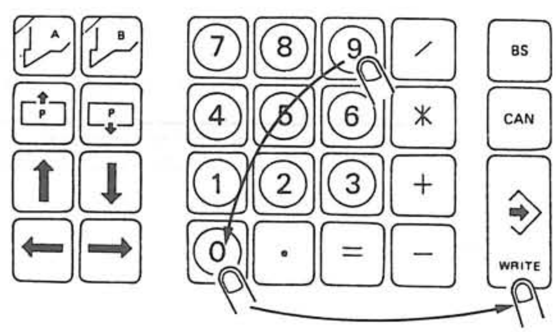
f) Define the start point (START PT).



Enter "5Ø" for "X" and "9Ø" for "Z".



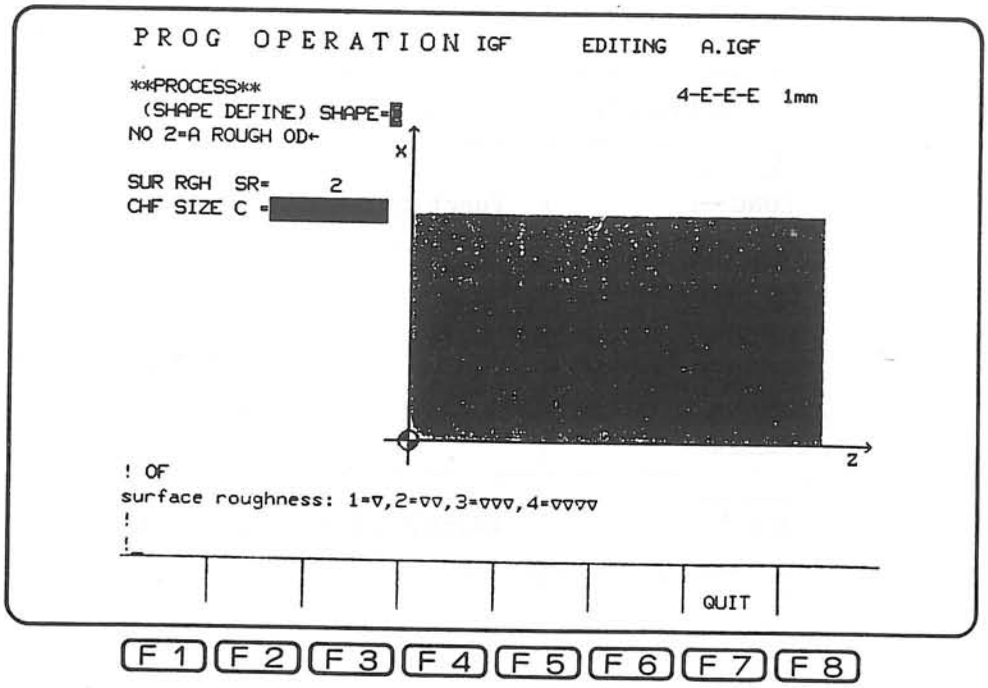
Entry of "5Ø"



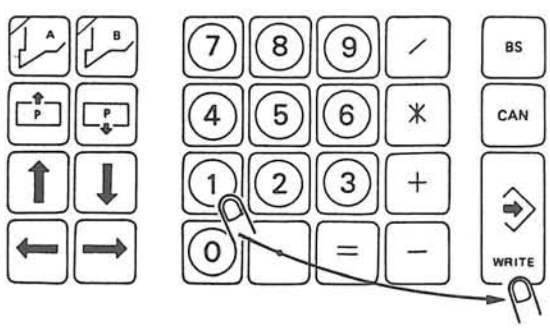
Entry of "9Ø"

The page is advanced to the next process.

The display should be as indicated below.



g) Enter "1" for chamfer size (CHF SIZE).



The procedure that follows is explained in the chart below. In this explanation, display pages are not provided since the pages shown in the previous data entry steps will be displayed repeatedly.

Step	Keys to be Pressed	Key Type	Remarks
h)	LONG ←	function key [F3]	SHAPE G= ←
	7 Ø WRITE		END PT Z=7Ø
i)	FACE ↑	function key [F1]	SHAPE G= ↑
	6 Ø WRITE		END PT X=6Ø
j)	TAPER ↖	function key [F2]	SHAPE G= ↖
	8 5 WRITE WRITE (2 times) 1 5 Ø WRITE		END PT X=85 For cursor advance (2 times) ANGLE A=15Ø°

Step	Keys to be Pressed	Key Type	Remarks
k)	EXTEND RADIUS 1 Ø WRITE	function key [F8] function key [F5]	SHAPE G= R R value (1Ø)
l)	LONG ← 3 Ø WRITE	function key [F3]	SHAPE G= ← END PT Z=3Ø
m)	EXTEND RADIUS 5 WRITE	function key [F8] function key [F5]	SHAPE G= R R values (5)
n)	FACE ↑ 1 Ø Ø WRITE	function key [F1]	SHAPE G= ↑ END PT X=1ØØ

Step	Keys to be Pressed	Key Type	Remarks
o)	EXTEND	function key [F8]	
p)	C-CHM 1 WRITE	function key [F6]	SHAPE G= C C=1

Shape defined in the steps above:

Compare Figs. 7-6-2-(1) and 7-6-2-(2).

a) New definition of shape:

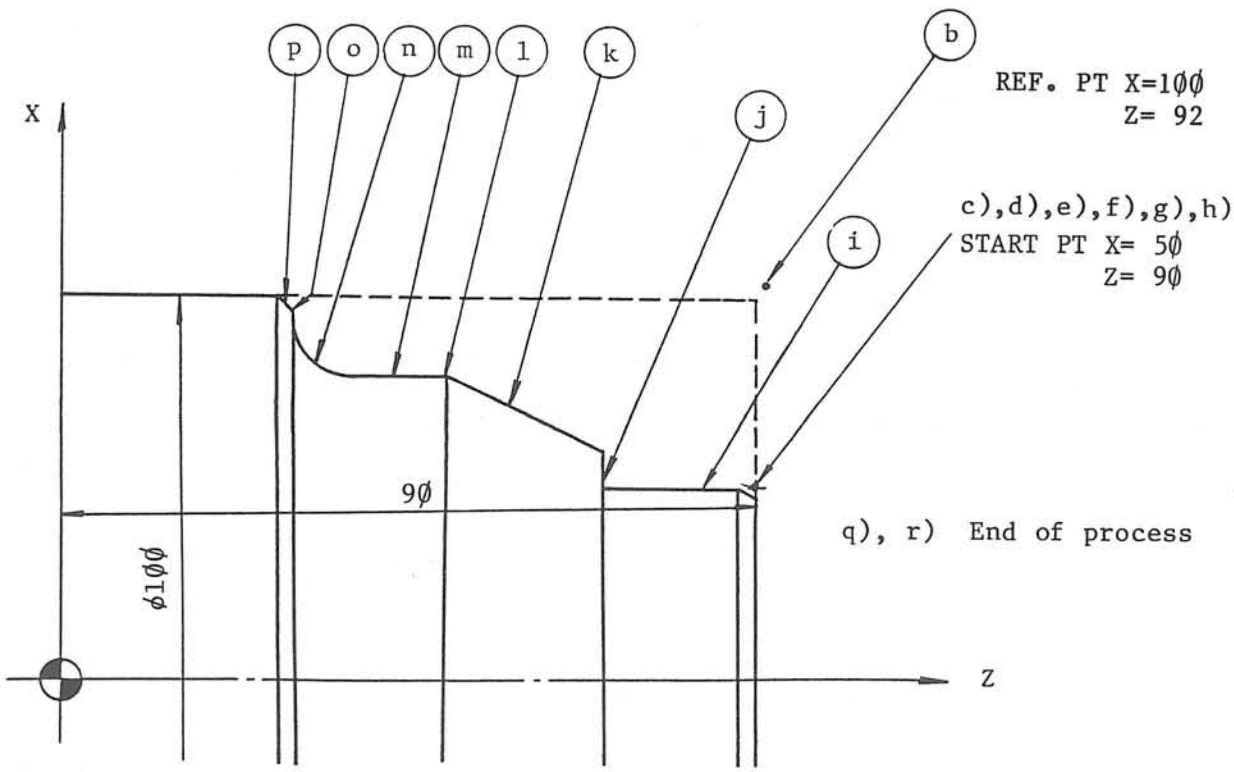
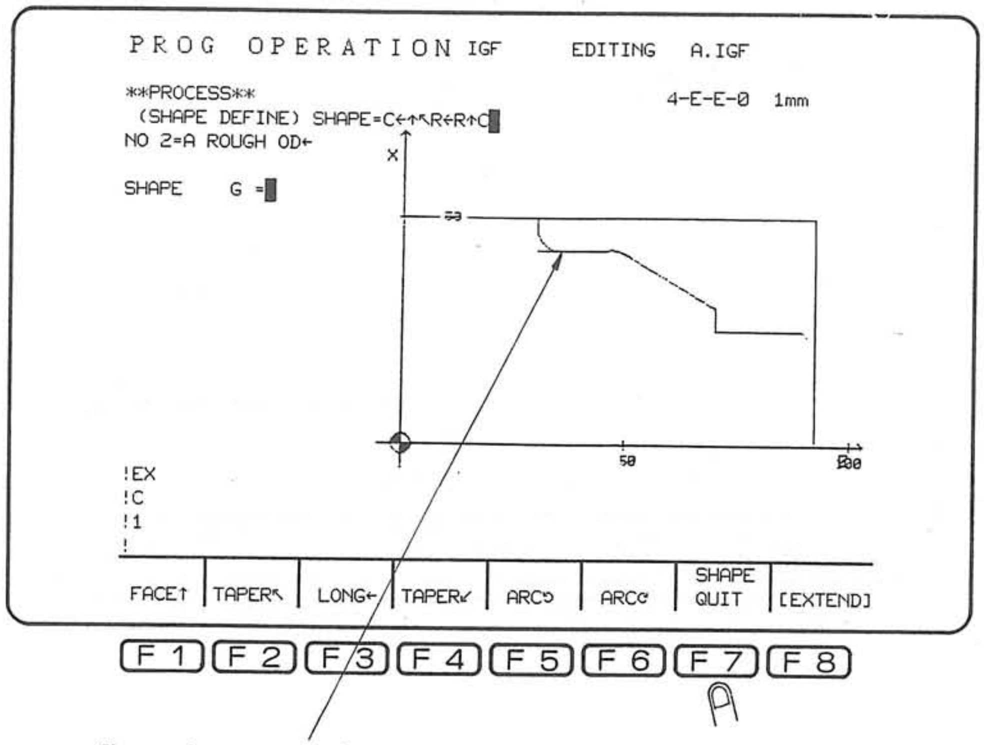


Fig. 7-6-2-(2)

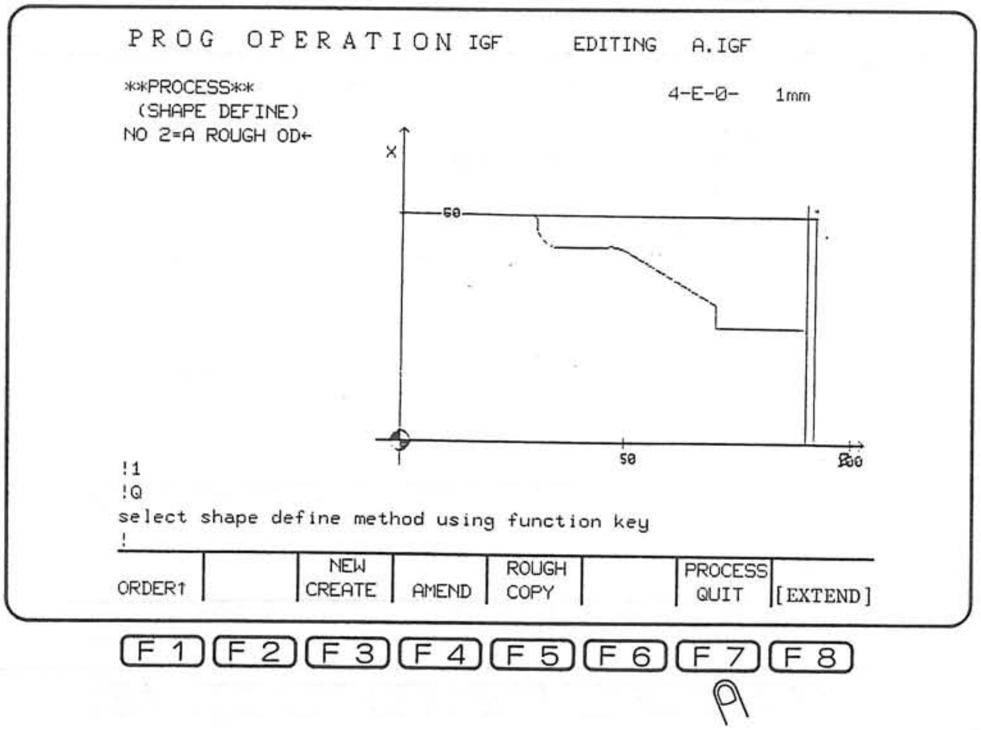
q) With the steps above, registration for ROUGH OD ← is completed. The display should be as indicated below.



Shape elements are displayed in the order they have been entered.

Press the function key [F7] (SHAPE QUIT). The page is returned to the first page of the PROCESS (SHAPE DEFINE).

r) The display should be as indicated below.



If no correction is required for the shape definition, press the function key [F7] (PROCESS QUIT).

This returns the display to the first page of the PROCESS.

This completes the registration of process NO 2 ROUGH OD←.

7-6-3. Registering Finish OD Face Process

Next process is the FINISH O.FACE ↓ registration.

The entry of the data can be made in the similar manner as registering the Rough OD Face and Rough OD processes. For some operations, display pages are not provided.

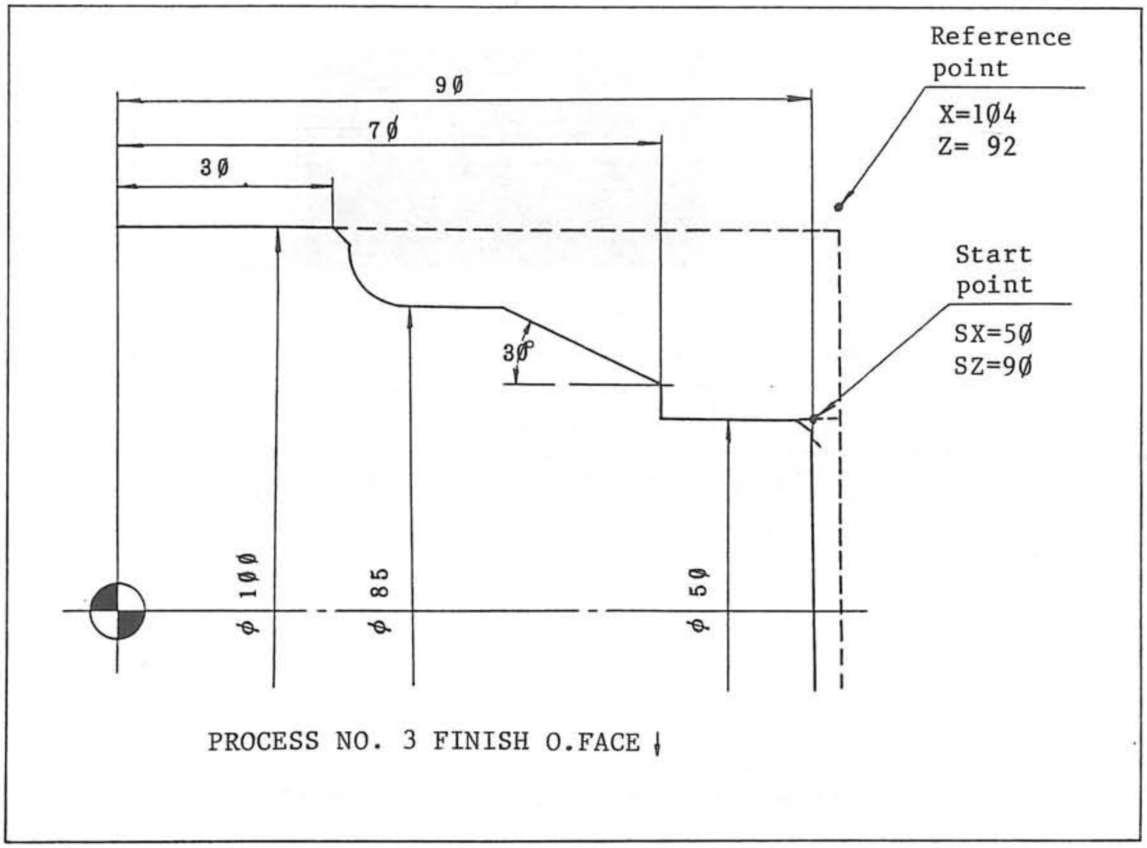
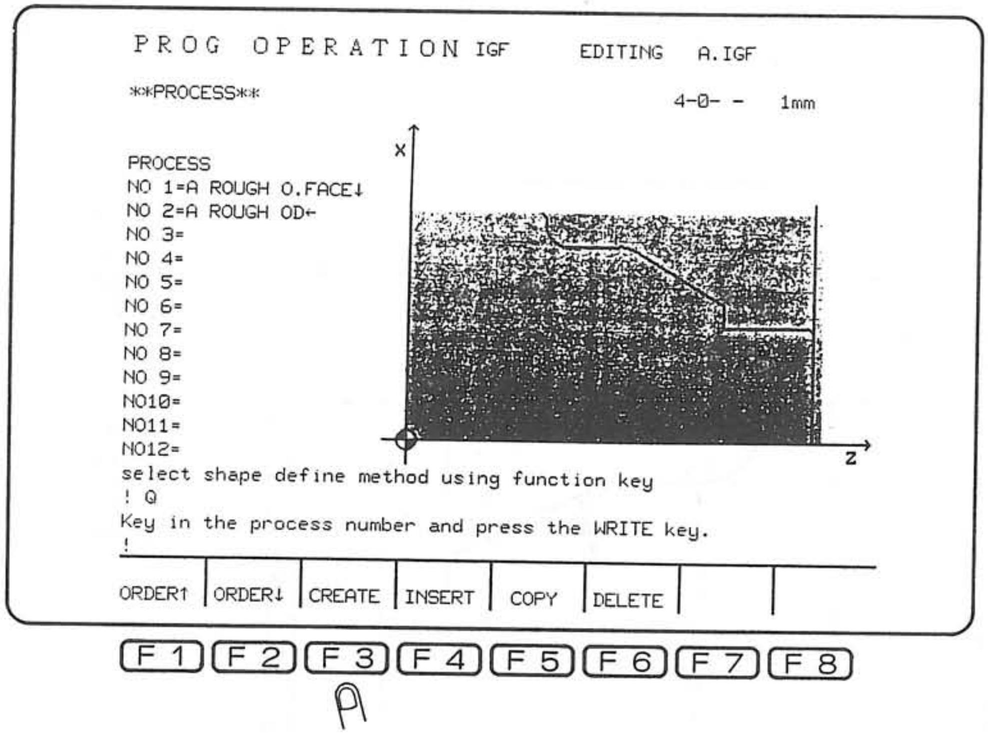


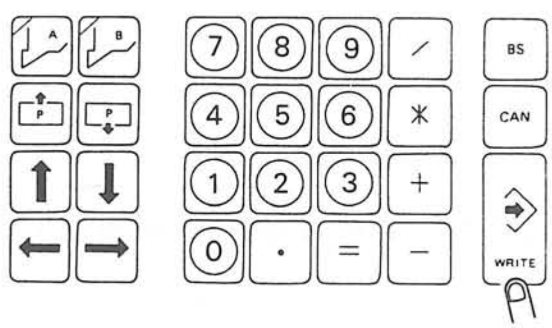
Fig. 7-6-3 Finish OD Facing Process

(1) PROCESS Designation (FINISH O.FACE ↓)

Enter the finish OD facing process next.



a) Press the function key [F3] (CREATE) and the WRITE key.



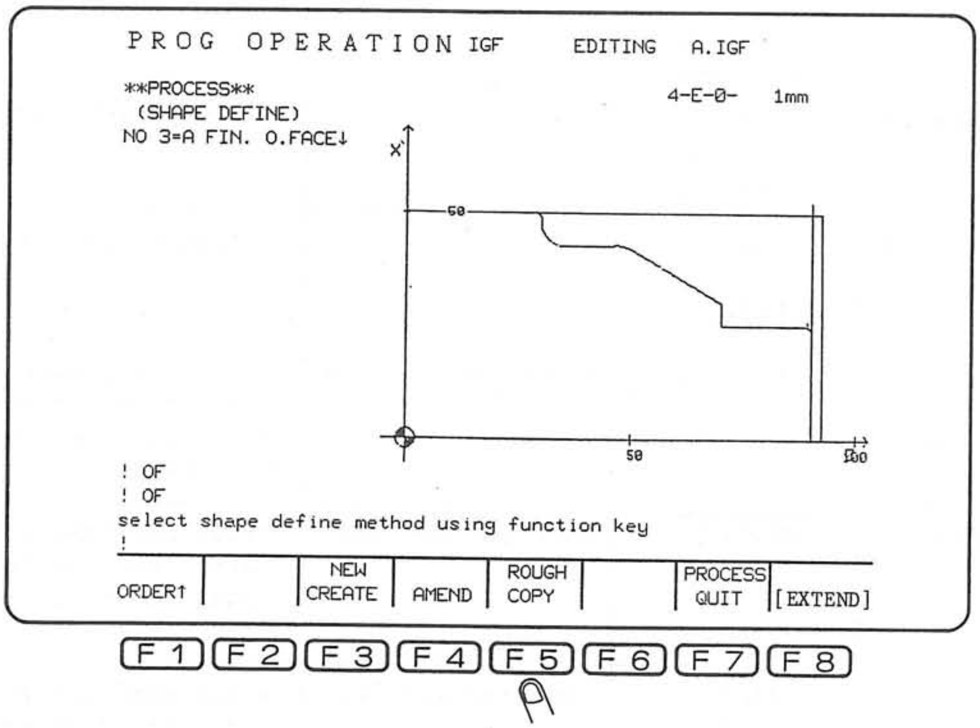
The processes that follow are 2) PROCESS (KINDS) designation, 3) PROCESS (DIRECTION) designation, 4) PROCESS (TOOL DATA NO) designation and 5) PROCESS (CONDITION) designation. These processes are summarized in the chart below:

PROCESS	Keys to be Pressed	Key Type	Remarks
(2) KINDS	3 WRITE		Select "3=FINISH".
(3) DIRECTION	3 WRITE		Select "3=OD FACE ↓".
(4) TOOL DATA	4 WRITE		Select "4 FINISH FACE ↓".
	ORDER ↓	function key [F2]	After confirming the correctness of the tool data displayed, press this key.
(5) CONDITION	ORDER ↓	function key [F2]	After confirming the correctness of the cutting condition data displayed, press this key.
	ORDER ↓	function key [F2]	After confirming the correctness of the cutting condition data displayed, press this key.

(6) PROCESS (SHAPE DEFINE) registration (FIN. O.FACE ↓)

The next process is the definition of the end face. This shape definition can be made in the same manner as defining rough OD facing process. However, the ROUGH COPY (function key [F6]) function will provide easier means for defining this shape.

a) Make sure that the PROCESS (SHAPE DEFINE) page is displayed on the CRT.

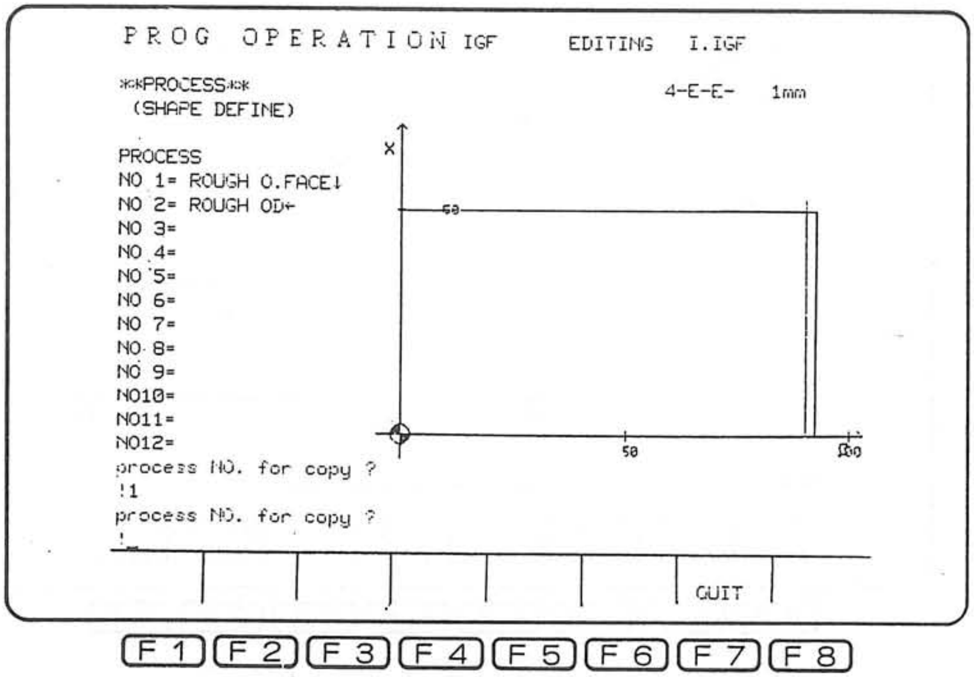


Press the function key [F5] (ROUGH COPY).

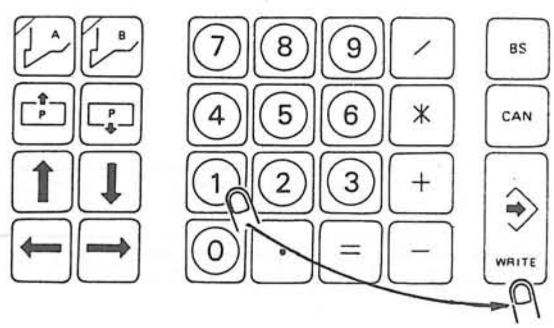
This displays the processes having been registered. Select the process to be copied from them.

In this example, since the shape to be defined is the finish OD facing, the shape defined in the rough OD facing should be copied. Note that the direction of cut must be the same for the process to be defined newly as the one to be copied. Therefore, the selectable process in this case is only the NO 1=ROUGH O.FACE ↓ .

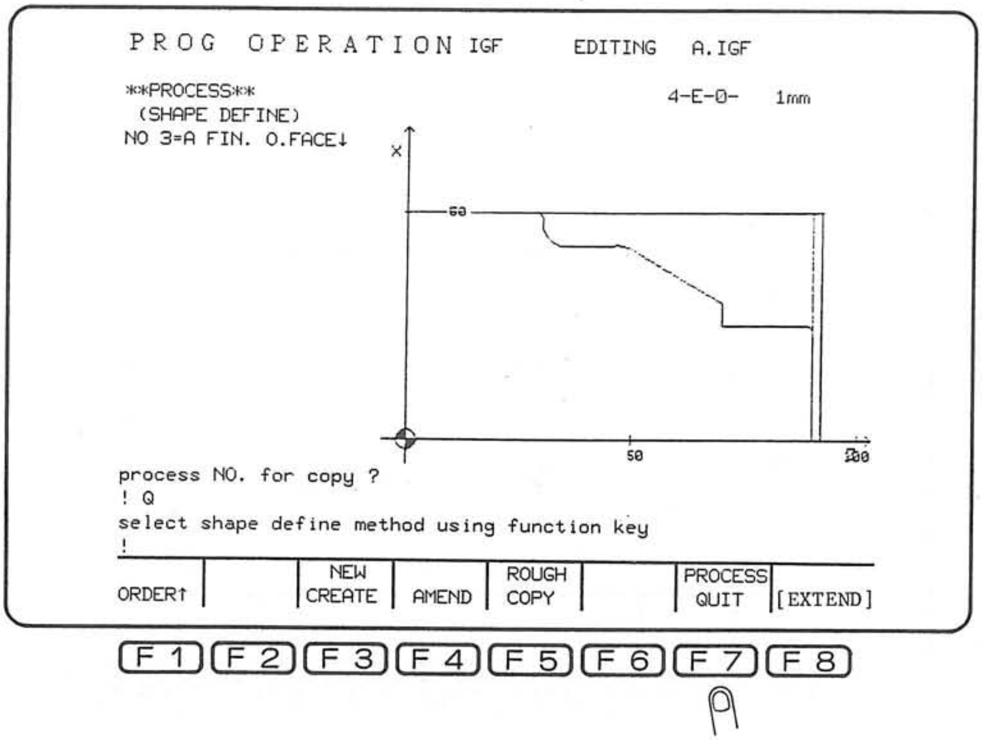
b) When the function key ROUGH COPY is pressed, the prompt "process NO. for copy?" will appear at the bottom line of the CRT.



To select the process "NO 1=ROUGH O.FACE ↓", enter "1".



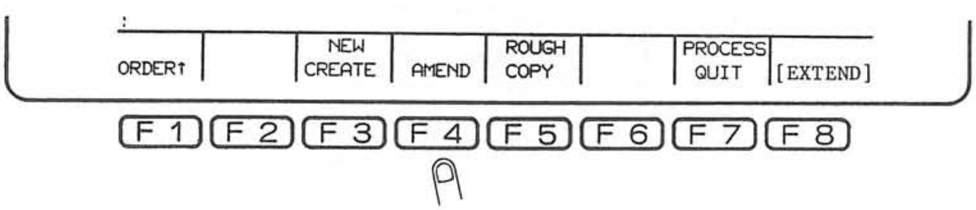
The shape copied is displayed on the CRT.



c) To end the rough copy function, press the function key [F7] (PROCESS QUIT).

d) If the shape copied requires edition, call the STEP AMEND function.

Press the function key [F4] (AMEND).



In this example, make corrections on the START PT SX data.

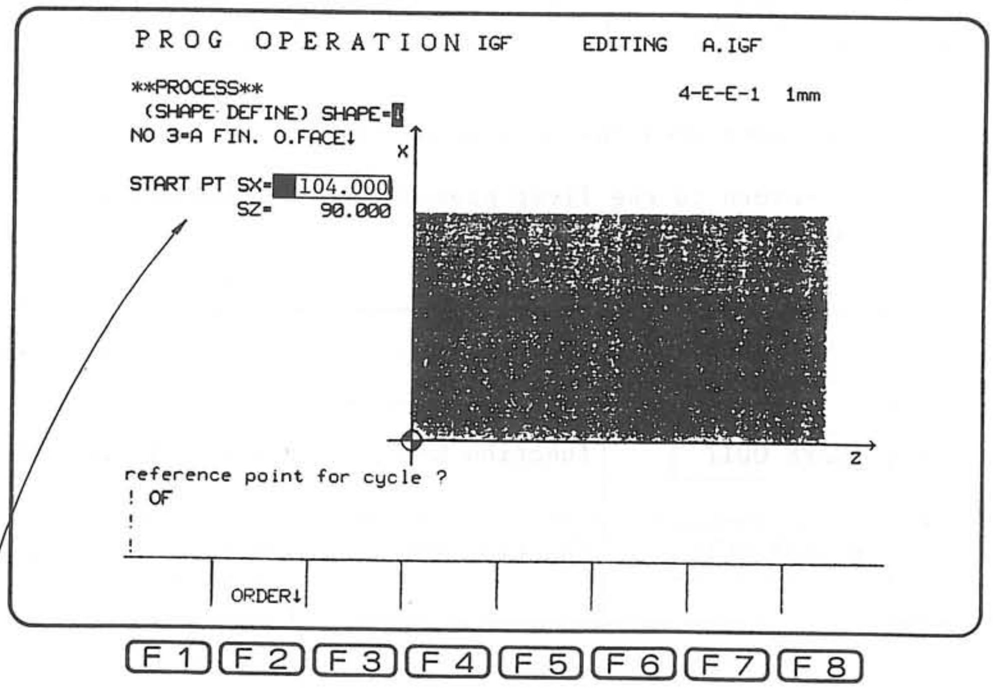
e) On the REF. PT data entry page, no edition is required.

Press the function key [F2] (ORDER↓).

f) The CRT shows SHAPE element page.

Press the WRITE key.

The display should be as indicated below.



The data in this column shows the data set in the ROUGH O.FACE↓ process registration.

Check the shape copied and edit the data as needed. In this example, START PT SX data is to be changed from SX=1Ø4 to SX=5Ø.

Follow the steps below:

<table>
<tr><th>Step</th><th>Keys to be Pressed</th><th>Key Type</th><th>Remarks</th></tr>
<tr><td>g) START PT SX</td><td>5 Ø
WRITE
ORDER ↓</td><td>

function key [F2]</td><td>Entry should be made with the cursor on SX=1Ø4.</td></tr>
<tr><td colspan="4">This completes the edition of the data.

To return to the first page of the PROCESS designation follow the steps below:</td></tr>
<tr><td>h)</td><td>QUIT</td><td>function key [F7]</td><td>End of SHAPE data entry</td></tr>
<tr><td>i)</td><td>SHAPE QUIT</td><td>function key [F7]</td><td>End of SHAPE element registration</td></tr>
<tr><td>j)</td><td>PROCESS QUIT</td><td>function key [F7]</td><td>End of SHAPE DEFINE process</td></tr>
</table>

7-6-4. Registering Finish OD Process

The FIN. OD ← process is registered next using the shape registered in the ROUGH OD process.

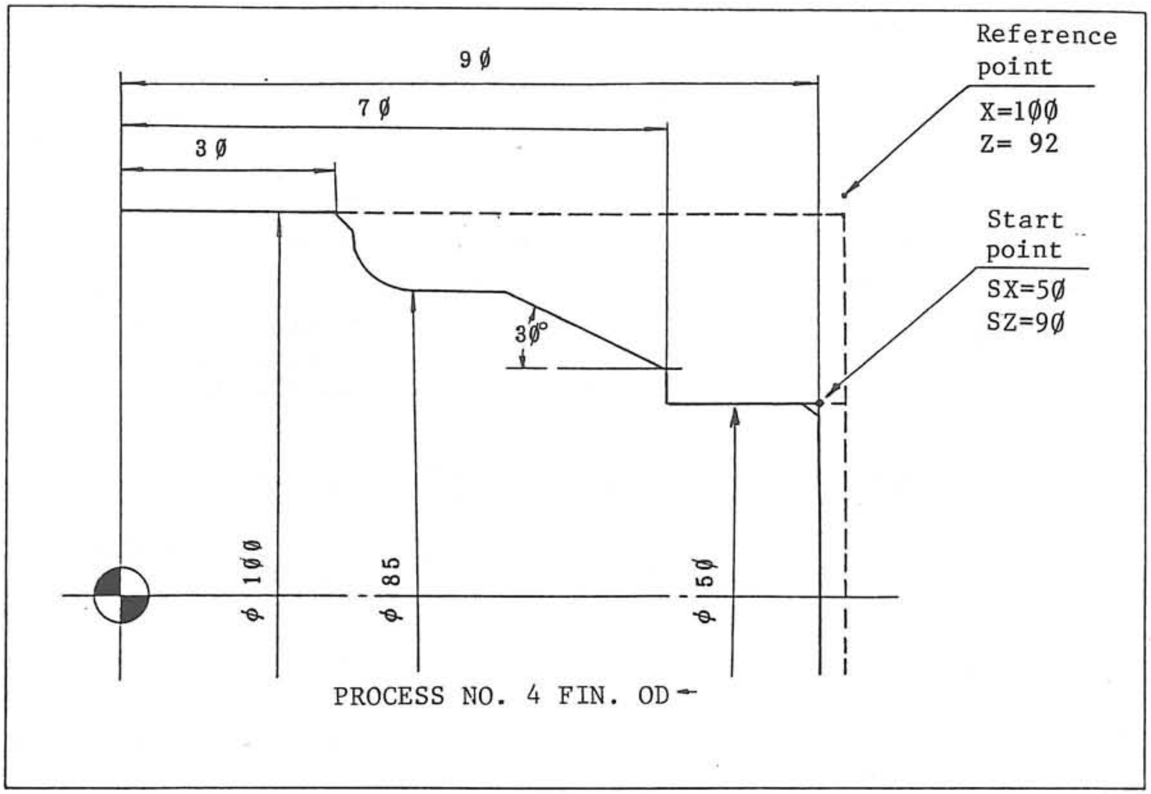


Fig. 7-6-4 Finish OD Turning Process

Follow the steps below:

PROCESS	Keys to be Pressed	Key Type	Remarks
(1) PROCESS	CREATE WRITE	function key [F3]	Registration of new process
(2) KINDS	3 WRITE		Select "3=FINISH".
(3) DIRECTION	1 WRITE		Select "1=OD←".

PROCESS	Keys to be Pressed	Key Type	Remarks
(4) TOOL DATA	3		Select "3 FINISH OD ←".
	WRITE		
	ORDER ↓	function key [F2]	After confirming the correctness of the tool data displayed, press this key.
(5) CONDITION	ORDER ↓	function key [F2]	After confirming the correctness of the cutting condition data displayed, press this key.
	ORDER ↓	function key [F2]	After confirming the correctness of the cutting condition data displayed, press this key.
(6) SHAPE DEFINE For defining the FIN. OD ← shape, copy the shape defined for ROUGH OD ← should be copied.			
a)	ROUGH COPY	function key [F5]	
b)	2 WRITE		Copy the shape defined in PROCESS NO 2=ROUGH OD ← for defining FIN. OD ← shape.
c)	QUIT	function key [F7]	The page returns to the first page of SHAPE DEFINE.
d)	PROCESS QUIT	function key [F7]	The page returns to the first page of PROCESS designation.

When editing of the shape element is not necessary, steps a) through d) can define the shape for FIN. OD ←.

7-6-5. Registering Grooving Process

Register the grooving process next.

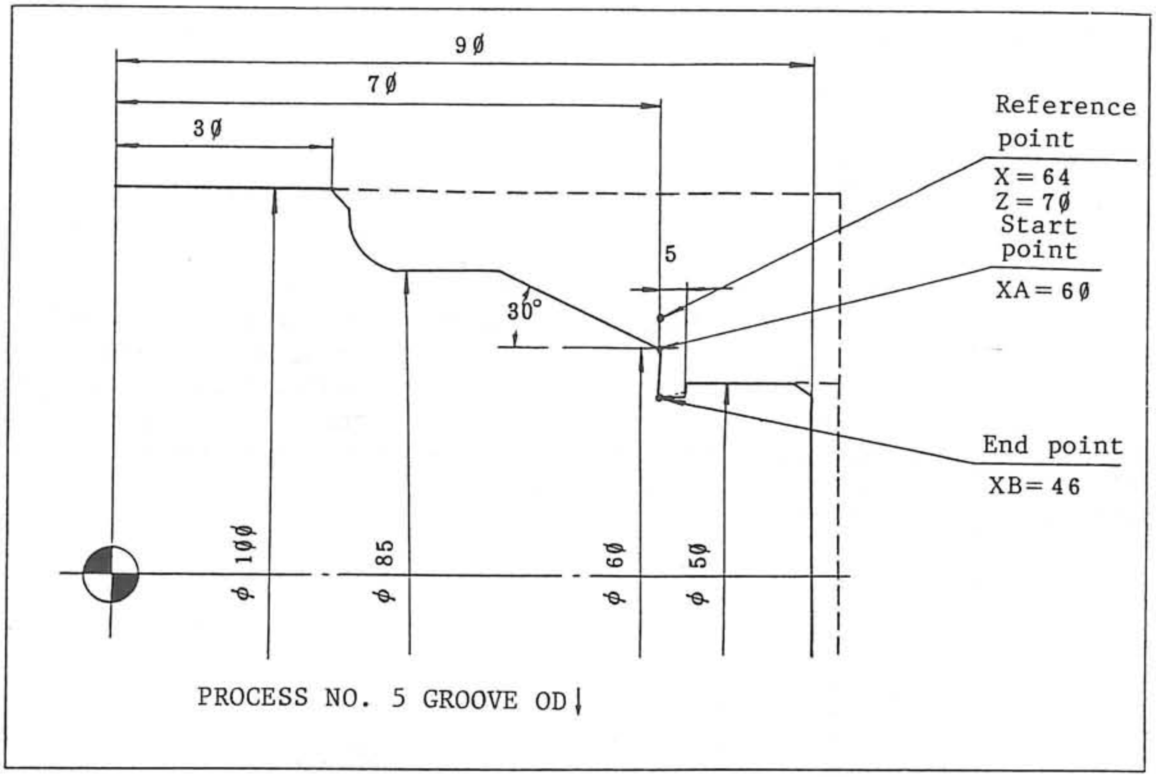


Fig. 7-6-5 Grooving Process

Follow the steps below:

PROCESS	Keys to be Pressed	Key Type	Remarks
(1) PROCESS	CREATE WRITE	function key [F3]	Registration of new process
(2) KINDS	5 WRITE		Select "5=GROOVE".
(3) DIRECTION	1 WRITE		Select "1=OD↓".

PROCESS	Keys to be Pressed	Key Type	Remarks
(4) TOOL DATA	9		Select "9 GROOVE OD ↓".
	WRITE		
	ORDER ↓	function key [F2]	After confirming the correctness of the tool data displayed, press this key.
(5) CONDITION	ORDER ↓	function key [F2]	After confirming the correctness of the cutting condition data displayed, press this key.
(6) SHAPE DEFINE	NEW CREATE	function key [F3]	
	6		REF. PT
	4		X=64
	WRITE		
	7		REF. PT
	Ø		XB=7Ø
	WRITE		
	ORDER ↓	function key [F2]	
	6		START PT
	Ø		XA=6Ø
	WRITE		
Either of these data entires can define the required shape.	4		START PT
	6		XB=46
	WRITE		
	(2		DEPTH DT=2 (DT in radius)
	WRITE)		
	SHAPE QUIT	function key [F7]	
	PROCESS QUIT	function key [F7]	Return to the first page of process designation.

7-6-6. Registering Thread Cutting Process

Register the thread cutting process next.

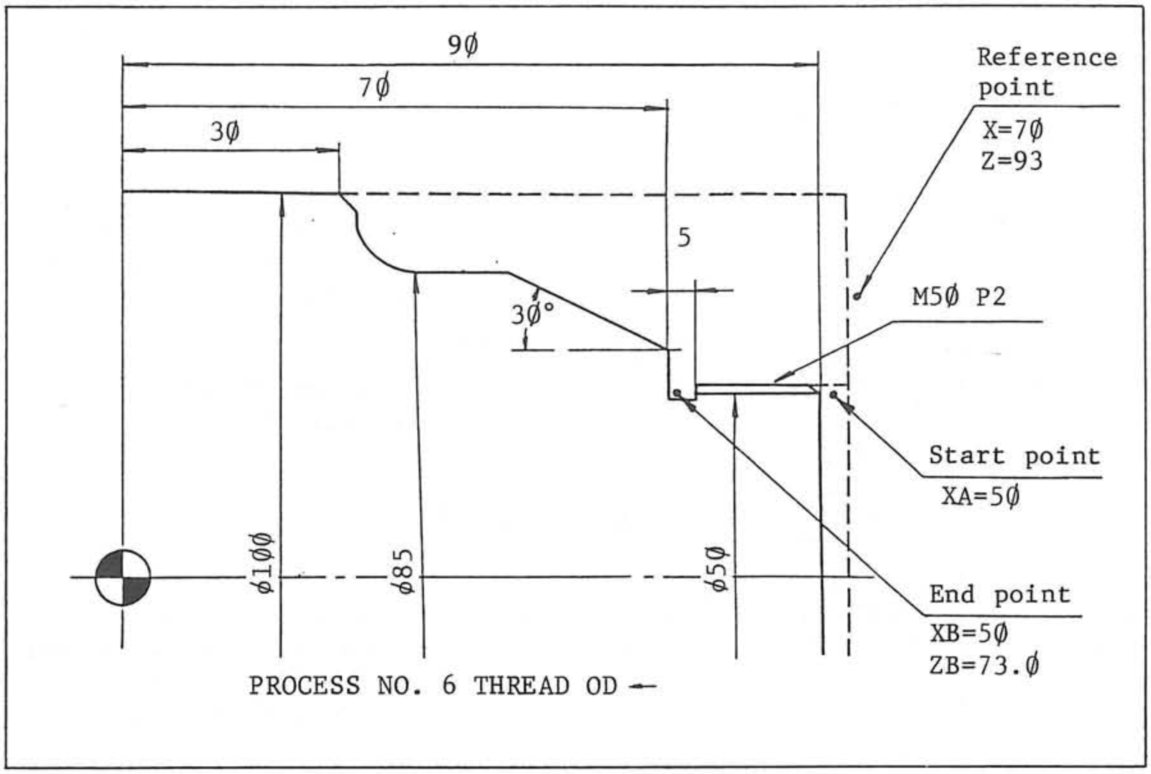


Fig. 7-6-6 Thread Cutting Process

Follow the steps below:

PROCESS	Keys to be Pressed	Key Type	Remarks
(1) PROCESS	CREATE WRITE	function key [F3]	Registration of new process
(2) KINDS	4 WRITE		Select "4=THREAD".
(3) DIRECTION	1 WRITE		Select "1=OD ←".

PROCESS	Keys to be Pressed	Key Type	Remarks
(4) TOOL DATA	7 WRITE		Select "7 THREAD OD←".
	ORDER ↓	function key [F2]	After confirming the correctness of the tool data displayed, press this key.
(5) CONDITION	2 WRITE ORDER ↓	function key [F2]	Enter thread pitch. (For "J", see Note 1.)
	ORDER ↓	function key [F2]	After confirming the correctness of the cutting condition data displayed, press this key.
(6) SHAPE DEFINE	NEW CREATE	function key [F3]	
	7 Ø WRITE 9 3 WRITE		REF. PT X=7Ø REF. PT Z=93 (allowance of 3 mm provided)
	ORDER ↓	function key [F2]	
	5 Ø WRITE		START PT XA=5Ø

PROCESS	Keys to be Pressed	Key Type	Remarks
	5 Ø WRITE 7 3 WRITE		END PT XB=5Ø END PT ZB=73.Ø
	1 . 3 WRITE		HEIGHT H=entry not necessary
	SHAPE QUIT	function key [F7]	Return to the first page of shape definition.
	PROCESS QUIT	function key [F7]	Return to the first page of process designation.

Note 1: By changing the values of F and J, switching of thread lead between inches and millimeters is possible.

<u>Metric</u>   F=2 (pitch)
J=1 (no. of threads per pitch)

→ Metric thread of pitch 2 mm

<u>Inches</u>   F=254 (pitch)
J=8 (no. of threads per pitch)

→ Inch thread of 8 threads/inch

Note 2: Infeed during thread cutting can be selectable from the following two methods.

1) Straight along thread face
2) Zig-zag

Note 3: Threads can be defined in two different methods:

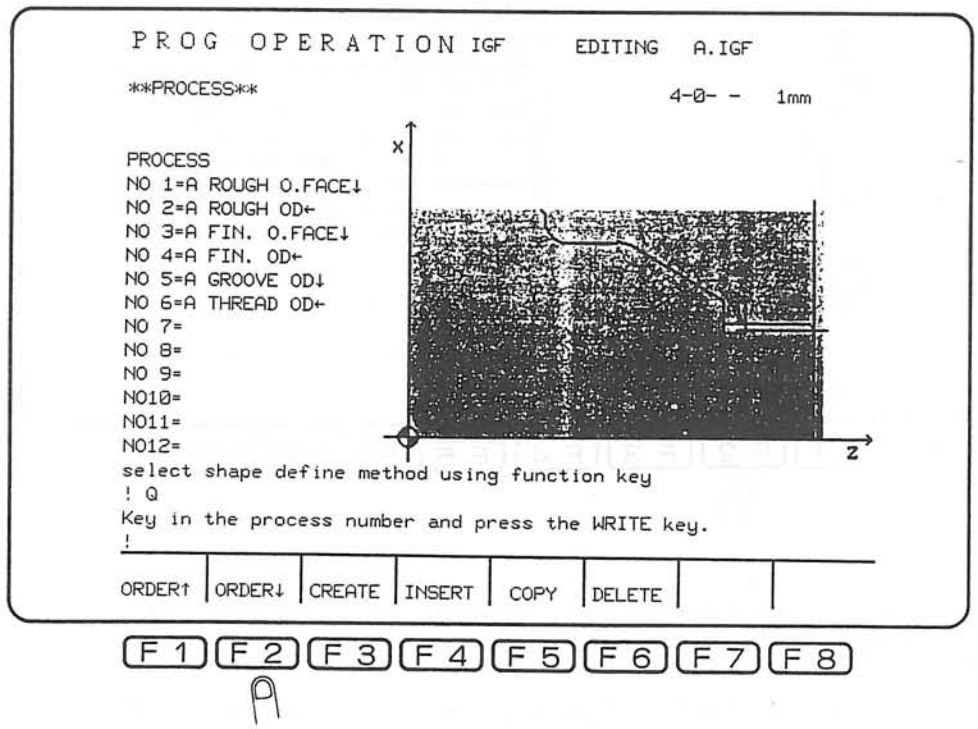
Root diameter
Nominal diameter

In this example, root diameter is used. For details of the thread cutting definition, refer to the Application Manual (Publication No. 2476-E).

7-7. CONFIRMATION OF REGISTERED PROCESSES

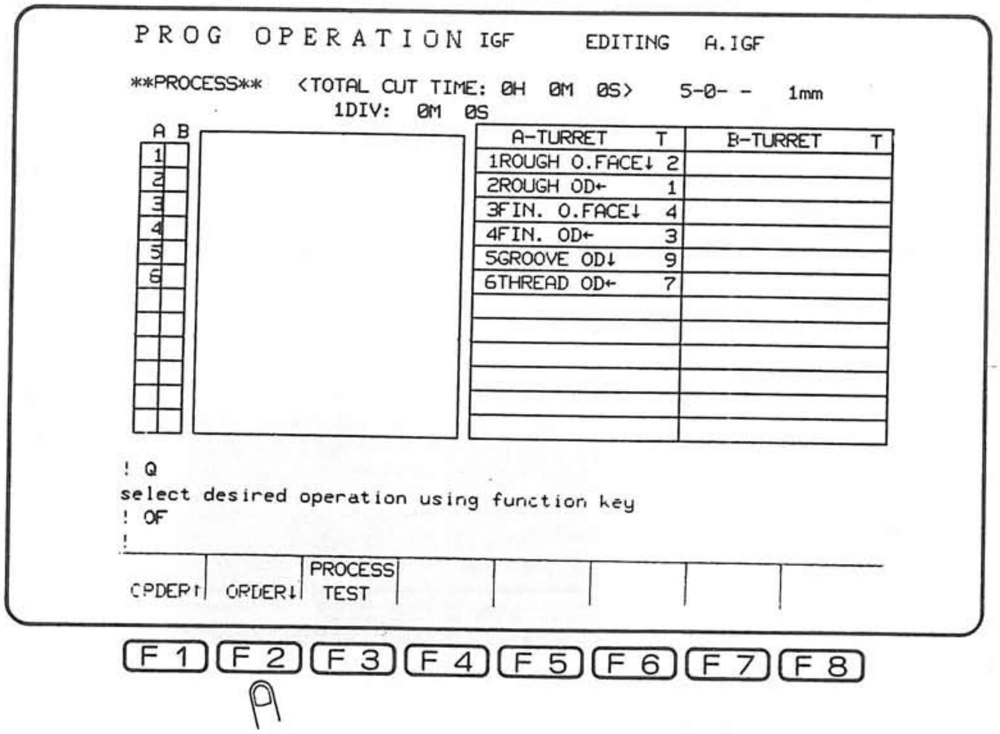
The processes having been registered are displayed on the CRT in the order they have been registered. Basically, the order displayed on the CRT is the order of machining.

1) The CRT should display the processes registered.



After the confirmation of the processes displayed, press the function key [F2] (ORDER ↓).

2) The CRT should display the PROCESS <TOTAL CUT TIME> page.

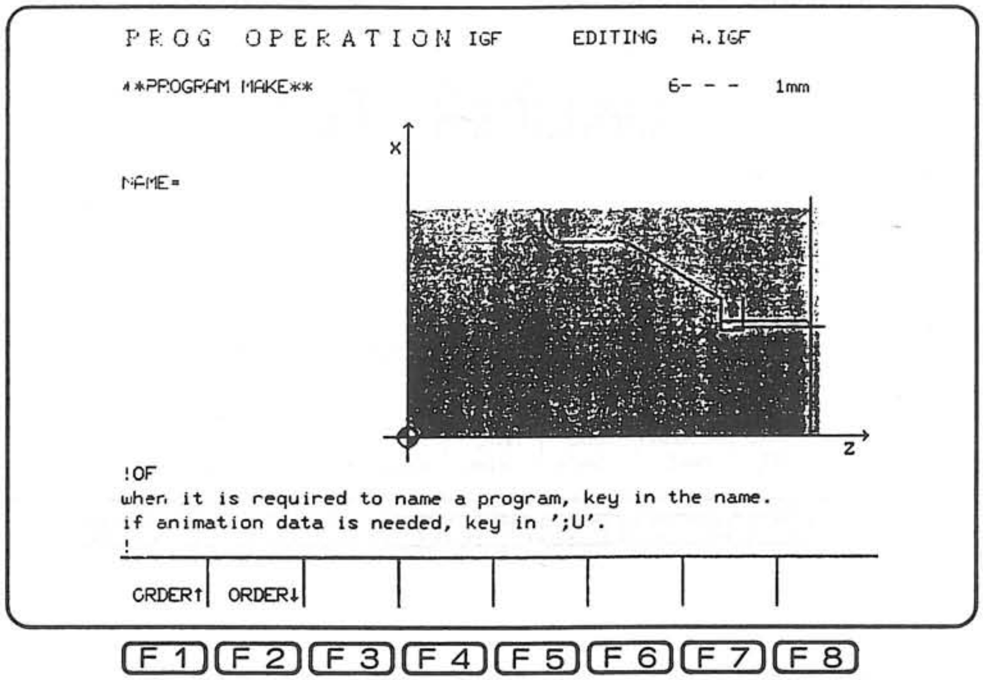


After the confirmation of the information displayed, press the function key [F2] (ORDER ↓).

If simulation or process change is required, press the function key [F3] (PROCESS TEST). For details of the simulation and the process change, refer to the Application Manual (Publication No. 2476-E).

7-8. MAKING NC PROGRAM

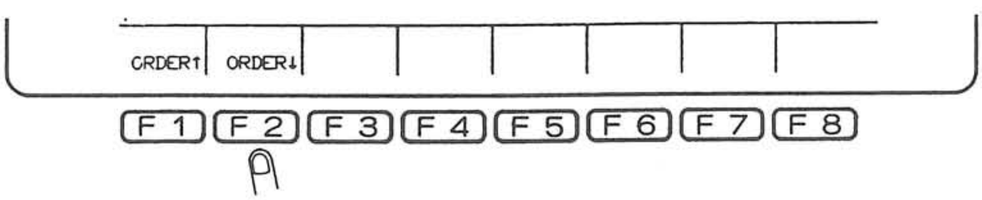
The page displayed should be the **PROGRAM MAKE**. From this page, IGF data can be compiled into an NC program and then stored into the bubble memory.



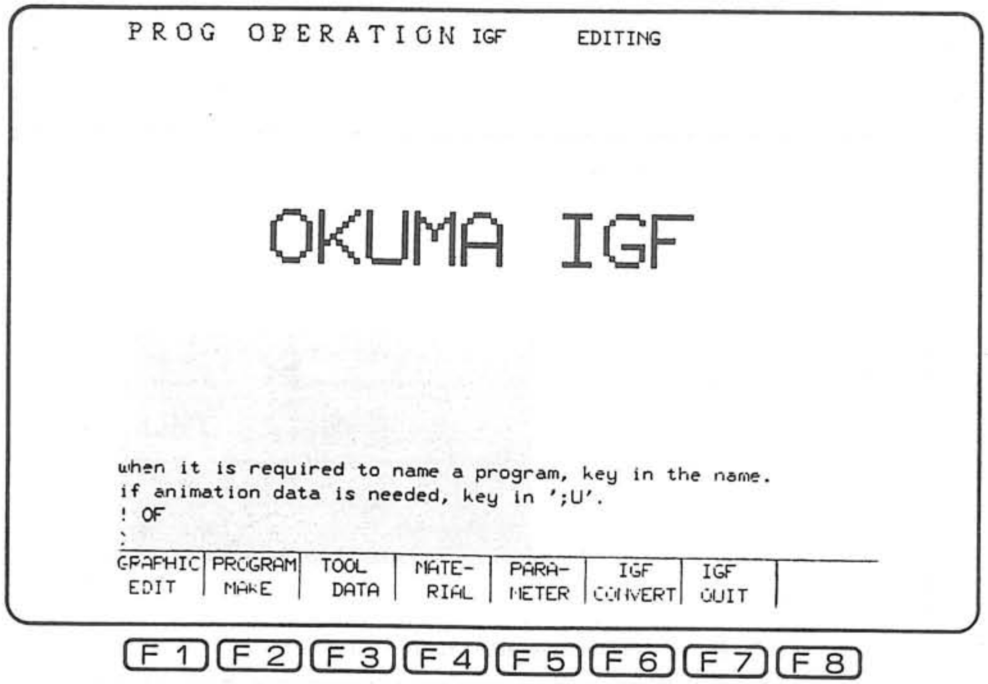
1) Key-in the NC program file name and the program name from the keyboard.

   In case the file name is SHAFT.MIN, key-in "S" "H" "A" "F" "T" ";" "U" from the keyboard and press the WRITE key.

   If the process is to terminate without making an NC program, press the function key [F2] (ORDER ↓).



2) The CRT is returned to the IGF start-up page.



Note: Input format of the file name and program name:

! <file-name>,<program-name> [WRITE]

For details, refer to Section 6, "Programming".

SECTION 8 MATH OPERATION FUNCTION FOR PARAMETER DATA INPUT

To set dimensions, feedrates, positions, etc. in the graphic edit operation, parameter setting operation, material data setting operation and tool data setting operation, arithmetic and function operation expressions can be used in addition to the conventional direct numerical data input method.

8-1. ARITHMETIC AND FUNCTION OPERATION SYMBOLS

Arithmetic and function operation expressions are designated using the symbols below.

Operation Symbol List

Symbol	Contents	Example	Remark
[+] [-]	Plus sign Minus sign	[+] 12.34 [[-] SIN 9Ø]	Usable only at the beginning of an expression or right after the "[" symbol.
[+] [-]	Addition Subtraction	12.3 [+] 456.7 [12.3 [-] 4] [-] [5 [-] 6]	
[*] [/]	Multiplication Division	12.34 [*] 56.7 [12.3 [/] 4] [/] 5.6	[*] indicates the multiplication symbol (x). [/] indicates the division symbol (÷).
[R] [S] [Q] [R] [T]	Square root	[R] (SP) 3Ø [S] [Q] [R] [T] [3Ø-2Ø]	If symbols [ ] are not used to indicate figures (operand) following the function symbol, place at least one "SP" between the function symbol and the operand.

Symbol	Contents	Example	Remark
$\boxed{S}$ $\boxed{S}\boxed{I}\boxed{N}$	Sine	$\boxed{S}$[45*2] $\boxed{S}\boxed{I}\boxed{N}$ SP 6Ø	- The figure written following the function symbol is an angle and expressed in units of degrees. - If symbols [ ] are not used to indicate figures (operand) following the function symbol, place at least one "SP" between the function symbol and the operand.
$\boxed{C}$ $\boxed{C}\boxed{O}\boxed{S}$	Cosine	$\boxed{C}$ SP 3Ø $\boxed{C}\boxed{O}\boxed{S}$[15+45]	
$\boxed{T}$ $\boxed{T}\boxed{A}\boxed{N}$	Tangent	$\boxed{T}$[45-15] $\boxed{T}\boxed{A}\boxed{N}$ SP [15*3/2]	
$\boxed{A}\boxed{S}$ $\boxed{A}\boxed{S}\boxed{I}\boxed{N}$	Arc sine	$\boxed{A}\boxed{S}$ SP Ø.5 $\boxed{A}\boxed{S}\boxed{I}\boxed{N}$[15.5/22.2]	- The result of operation is an angle in units of degrees. - If symbols [ ] are not used to indicate figures (operand) following the function symbol, place at least one "SP" between the function symbol and the operand.
$\boxed{A}\boxed{C}$ $\boxed{A}\boxed{C}\boxed{O}\boxed{S}$	Arc cosine	$\boxed{A}\boxed{C}$[Ø.8*Ø.6] $\boxed{A}\boxed{C}\boxed{O}\boxed{S}$ SP Ø.45	
$\boxed{A}\boxed{T}$ $\boxed{A}\boxed{T}\boxed{A}\boxed{N}$	Arc tangent (-9Ø° - 9Ø°)	$\boxed{A}\boxed{T}$ SP 45 $\boxed{A}\boxed{T}\boxed{A}\boxed{N}$[45*2]	- The result of operation is an angle in units of degrees. - If symbols [ ] are not used to indicate figures (operand) following the function symbol, place at least one "SP" between the function symbol and the operand.

Symbol	Contents	Example	Remark
$\boxed{\text{A}}\boxed{\text{T}}\boxed{\text{2}}$ $\boxed{\text{A}}\boxed{\text{T}}\boxed{\text{A}}\boxed{\text{N}}\boxed{\text{2}}$	Arc tangent (-18Ø° - 18Ø°)	$\boxed{\text{A}}\boxed{\text{T}}\boxed{\text{2}}$[Ø.5, 1.5] $\boxed{\text{A}}\boxed{\text{T}}\boxed{\text{A}}\boxed{\text{N}}\boxed{\text{2}}$[Ø.45, 1]	- Designation should always be AT2 [numerator, denominator]. - Symbols [ ] cannot be omitted.

Remarks:

1. If the expression does not follow the format indicated above, or an overflow occurs in the course of operation, a calculation error occurs.

   Example: S3Ø → Calculation error (No SP between "S" and "3Ø")

2. As indicated in the list above, symbols [ ] can be used for operation. Although the nesting level for the usage of them is not specially limited, fourth nesting level is the maximum depth to guarantee the results of operations.

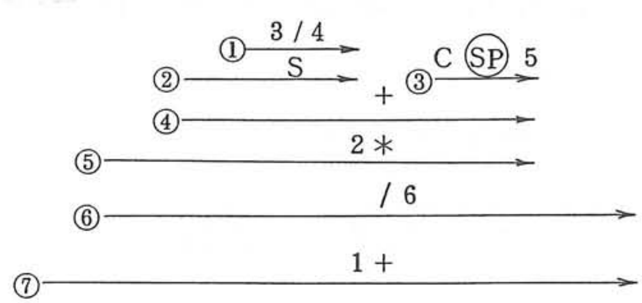
   Example: R SP [18Ø+[S*[3Ø-5]-2Ø*[6-2]]]

3. Calculation order follows usual arithmetic operation rules:

   In parentheses → Functions → Multiplication/division → Addition/subtraction

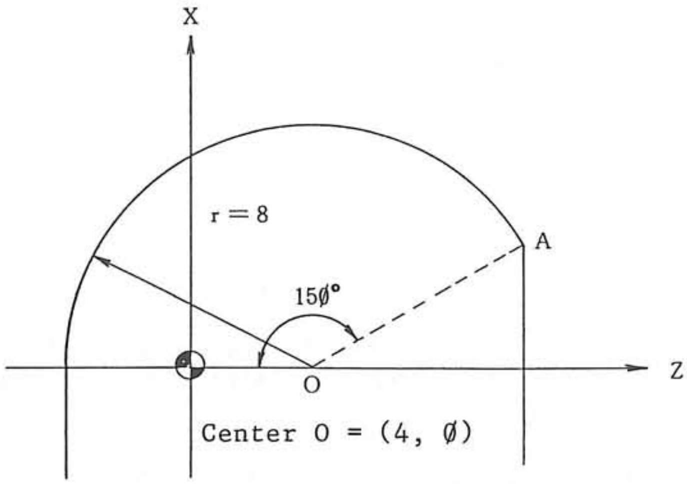
   If operations of the same calculation priority are used, calculation is made from the leftmost operations in order.

   Example: 1+2*[S [3/4] + C SP 5]/6



8-2. INPUT EXAMPLES

To input the position data of point A in the figure given at the right, the use of an expression will simplify data input process as well as provide accuracy.



Point A $\begin{cases} Z = 4 + 8 \times \cos(18Ø - 15Ø) \\ X = 8 \times \sin(18Ø - 15Ø) \end{cases}$

(1) Locate the cursor at the Z-coordinate input position for point A.

(2) Input the expression.

For the Z-coordinate of point A, input as follows:

[4] [+] [8] [*] [C] [[] [1] [8] [Ø] [-] [1] [5] [Ø] []] [WRITE]

The input is displayed at the console area of the CRT screen as 4+8*C[18Ø-15Ø]. The result of calculation is displayed at the data column.

(3) Locate the cursor at the X-coordinate input position for point A.

(4) Input the expression.

For the X-coordinate of point A, input as follows:

[8] [*] [S] [[] [1] [8] [Ø] [-] [1] [5] [Ø] []] [WRITE]

The input is displayed at the console area of the CRT screen as 8*S[18Ø-15Ø]. The result of the calculation is displayed at the data column.

Note: Input of symbol $\boxed{[}$ ..... Press $\boxed{T^{[}}$ while holding down the UPPER CASE key.

Input of symbol $\boxed{]}$ ..... Press $\boxed{U^{]}}$ while holding down the UPPER CASE key.

SECTION 9 DATA MODIFICATION USING ADDITION FUNCTION

The function to modify the cursor located data by adding or subtracting the keyed in figure to or from the current data is supported to facilitate data editing in the graphic edit operation.

Data Input Procedure:

Input the data in the following manner.

(1) Locate the cursor on the data for which keyed in data is to be added.

(2) Key in the data in the following format:

[A] [D] (SP) figure or expression [WRITE]

The keyed in data or the result of calculation is added to the data which is indicated by the cursor.

Note 1: If the keyed in data or the result of calculation is negative, subtraction is conducted.

Note 2: If data input is made for the column where no data has been set, the keyed in data or the result of calculation is set as is.

Note 3: For details of expression, refer to Section 8.

Data Input Examples:

[A] [D] [SP] [1] [5] .... Figure "15" is added to the current cursor located data.

[A] [D] [SP] [-] [Ø] [.] [1] ...... Figure "Ø.1" is subtracted from the current cursor located data.

[A] [D] [SP] [Ø] [.] [1] [*] [3] ... Figure "Ø.3" is added to the current cursor located data.

[A] [D] [SP] [-] [1] [/] [5] ...... Figure "Ø.2" is subtracted from the current cursor located data.

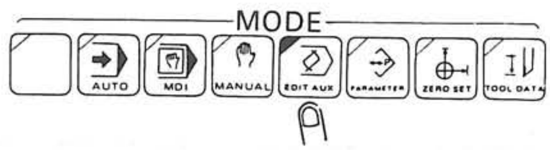
SECTION 1Ø CONVERSION FUNCTION

The conversion function converts the LAP function, nose R compensation function and compound fixed cycle function programmed using the IGF into the general program for format.

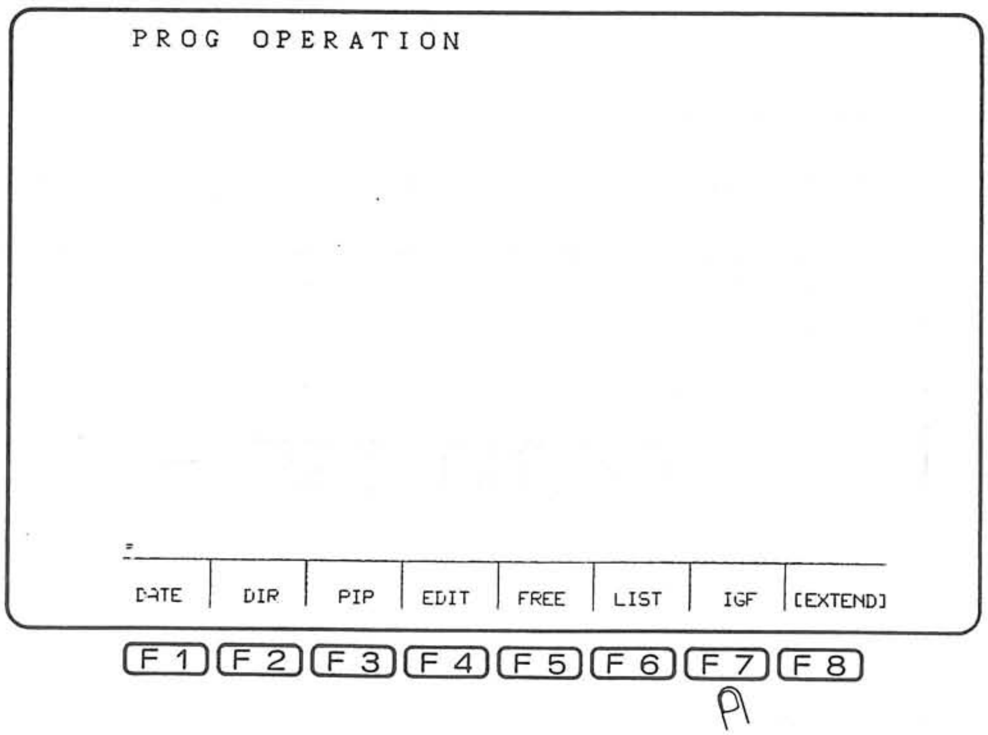
Although this function can convert the NC program made without using the IGF, the program after the conversion cannot be guaranteed.

1Ø-1. OPERATION PROCEDURE

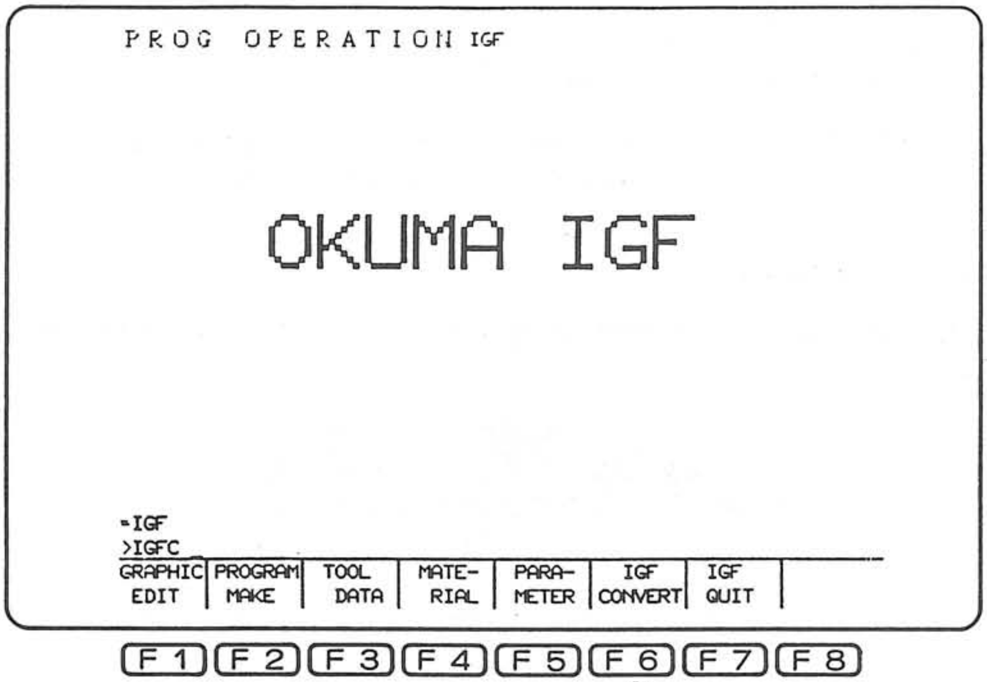
1) Select the PROGRAM OPERATION mode by pressing the EDIT AUX. key.



2) Press the function key [F7] (IGF) to select the IGF mode.

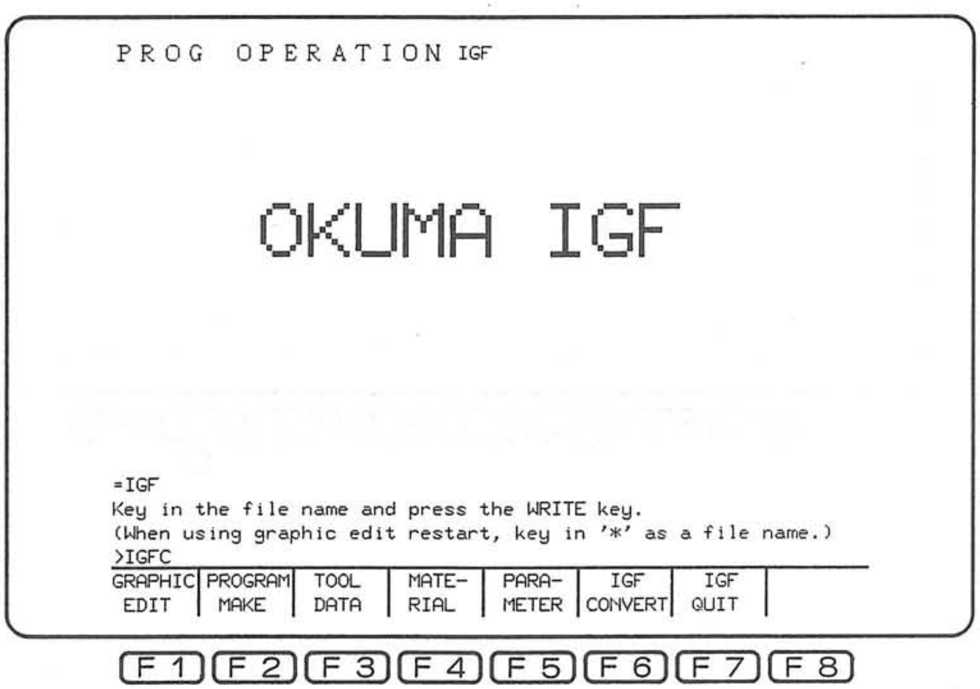


The IGF start-up page will be displayed.



3) Press the function key [F6] (IGF CONVERT).

The prompt ">IGFC" will appear at the lower area of the CRT.



4) Key in the input file name, output file name and option parameter from the keyboard.

>IGFC <input-file-name>(,<output-file-name>)(;<option-parameter>)

Then press the WRITE key.

Note that the entry of information in ( ) can be omitted.

Option Parameter

Code	Contents	When Omitted
A	The program of the designated input file name is converted into the program for the control* and stored in the bubble memory with an output file name assigned.	Conversion into the program for the control*
B	The program of the designated input file name is converted into the program for the OSP3ØØØL and stored in the bubble memory with an output file name assigned.	Conversion into the program for the control*
S	The output program is tabulated with spaces inserted.	Spaces are eliminated from the output program.
T	If the input program is a subprogram, it is converted into the program for B-turret and stored in the bubble memory with output file name assigned.	Conversion into program for A-turret

* OSP5ØØL-G/OSP5ØØØL-G without LAP, nose R compensation and compound fixed cycle functions

Example: For converting the file A.MIN into the program for OSP3ØØØL (file name B.MIN) with spaces inserted

>IGFC A.MIN,B.MIN;BS [WRITE]

or

>IGFC ,B;BS [WRITE]

5) After the completion of the conversion, the prompt ">" will be displayed on the CRT.

1Ø-2. CONVERSION CONTENTS

1Ø-2-1. LAP

(1) G Codes

G8Ø	End of contour definition
G81	Start of longitudinal contour definition
G82	Start of transverse (on end face) contour definition
G85	Bar turning cycle
G86	Copy turning cycle
G87	Finish cut cycle

(2) LAP Parameters

D	Depth of cut for roughing
E	Feedrate for roughing along the shape

(3) Example of Conversion of G85 (bar turning cycle), G87 (finish cut cycle) and shape definition (G81 - G8Ø)

Tool paths generated in the cycles called by the G85 and G87 commands (Fig. 1Ø-1) are converted into a general program.

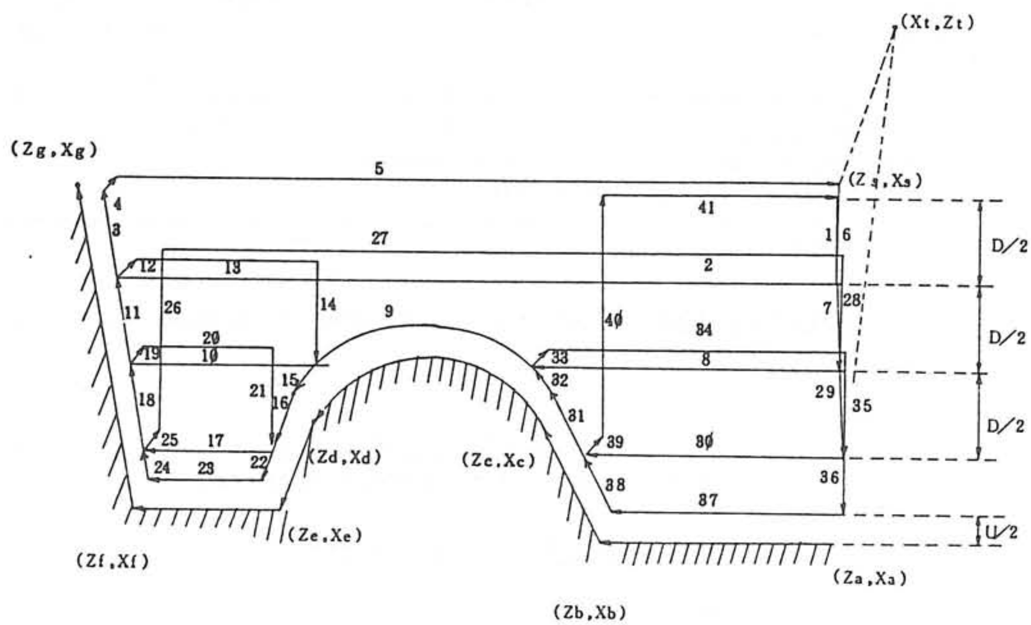


Fig. 1Ø-1 Tool Paths in Roughing and Finishing Cycles

<Input Program>

```
NØØØ1 GØØ Xt Zt S T M
NØØØ2     Xs Zs
NØØØ3 G85 NØØØ4 D F U W     *1
NØØØ4 G81
NØØØ5 GØ1 Xa Za Ea
NØØØ6     Xb Xb Eb
NØØØ7     Xc Zc
NØØØ8 GØ3 Xd Zd Id Kd       *2
NØØØ9 GØ1 Xe Ze
NØØ1Ø     Xf Zf
NØØ11     Xg Zg Eg
NØØ12 G8Ø
NØØ13 GØØ Xt Zt S T M
NØØ14     Xs Zs
NØØ15 G87 NØØ16             *3
NØØ16 G81
NØØ17 GØ1 Xa Za Fa
NØØ18     Xb Zb
NØØ19     Xc Zc
NØØ2Ø GØ3 Xd Zd Id Kd Fd    *2
NØØ21 GØ1 Xe Ze Fe
NØØ22     Xf Zf
NØØ23     Xg Zg Fg
NØØ24 8Ø

          :
          :
```

*1: Call of rough turning cycle
*2: Shape definition
*3: Call of finish turning cycle

<Output Program>

```
NØØØ1 GØØ Xt  Zt  S T M
NØØØ2     Xs  Zs
NØØØ3 GØ1 X1  Z1  Fa              (1)
NØØØ4     X2  Z2  F               (2)
NØØØ5     X3  Z3  Fg              (3)
NØØØ6 GØØ X4  Z4                  (4)
NØØØ7     X5  Z5                  (5)
NØØØ8     X6  Z6                  (6)
NØØØ9 GØ1 X7  Z7  Fa              (7)
NØØ1Ø     X8  Z8  F               (8)
NØØ11 GØ3 X9  Z9  I9 K9 Fb        (9)
NØØ12 GØ1 X1Ø Z1Ø F               (1Ø)
NØØ13     X11 Z11 Fg              (11)
NØØ14 GØØ X12 Z12                 (12)
NØØ15     X13 Z13                 (13)
NØØ16     X14 Z14                 (14)
NØØ17 GØ3 X15 Z15 I15 K15 Fb      (15)
NØØ18 GØ1 X16 Z16                 (16)
NØØ19     X17 Z17 F               (17)
NØØ2Ø     X18 Z18 Fg              (18)
NØØ21 GØØ X19 Z19                 (19)
NØØ22     X2Ø Z2Ø                 (2Ø)
NØØ23     X21 Z21                 (21)
NØØ24 GØ1 X22 Z22 Fb              (22)
NØØ25     X23 Z23                 (23)
NØØ26     X24 Z24 Fg              (24)
NØØ27 GØØ X25 Z25                 (25)
NØØ28     X26 Z26                 (26)
NØØ29     X27 Z27                 (27)
NØØ3Ø     X28 Z28                 (28)
NØØ31 GØ1 X29 Z29 Fa              (29)
NØØ32     X3Ø Z3Ø F               (3Ø)
NØØ33     X31 Z31 Fb              (31)
NØØ34 GØ3 X32 Z32 I32 K32         (32)
NØØ35 GØØ X33 Z33                 (33)
NØØ36     X34 Z34                 (34)
NØØ37     X35 Z35                 (35)
NØØ38 GØ1 X36 Z36 Fa              (36)
NØØ39     X37 Z37 Fb              (37)
NØØ4Ø     X38 Z38                 (38)
NØØ41 GØØ X39 Z39                 (39)
NØØ42     Xs                      (4Ø)
NØØ43         Zs                  (41)
NØØ44     Xt  Zt  S T M
NØØ45     Xs  Zs
NØØ46 GØ1 Xa  Za  Fa
NØØ47     Xb  Zb
NØØ48     Xc  Zc
NØØ49 GØ3 Xd  Zd  Id Kd Fd
NØØ5Ø GØ1 Xe  Ze  Fe
NØØ51     Xf  Zf
NØØ52     Xg  Zg  Fg
          :
```

(4) Example of Conversion of G86 (copy turning cycle) and shape definition (G81 - G8Ø)

Tool paths generated in the cycles called by the G86 command (Fig. 1Ø-2) are converted into a general program.

<Input Program>

```
NØØØ1 GØØ Xt Zt S T M
NØØØ2     Xs Zs
NØØØ3 G86 NØØØ4 D F U W
NØØØ4 G81
NØØØ5 GØ1 Xa Za Ea
NØØØ6     Xb Xb Eb
NØØØ7     Xc Zc Ec
NØØØ8 G8Ø

      :
      :
```

<Output Program>

```
NØØØ1 GØØ Xt  Zt S T M
NØØØ2     Xs  Zs
NØØØ3 GØ1 X1  Z1  Fa      (1)
NØØØ4     X2  Z2  Fb      (2)
NØØØ5     X3  Z3  Fc      (3)
NØØØ6 GØØ     Zs          (4)
NØØØ7 GØ1 X5  Z5  Fa      (5)
NØØØ8     X6  Z6  Fb      (6)
NØØØ9     X7  Z7  Fc      (7)
NØØ1Ø GØØ     Zs          (8)
NØØ11 GØ1 X9  Z9  Fa      (9)
NØØ12     X1Ø Z1Ø Fb      (1Ø)
NØØ13     X11 Z11 Fc      (11)
NØØ14 GØØ     Zs          (12)
NØØ15     Xs              (13)

          :
          :
```

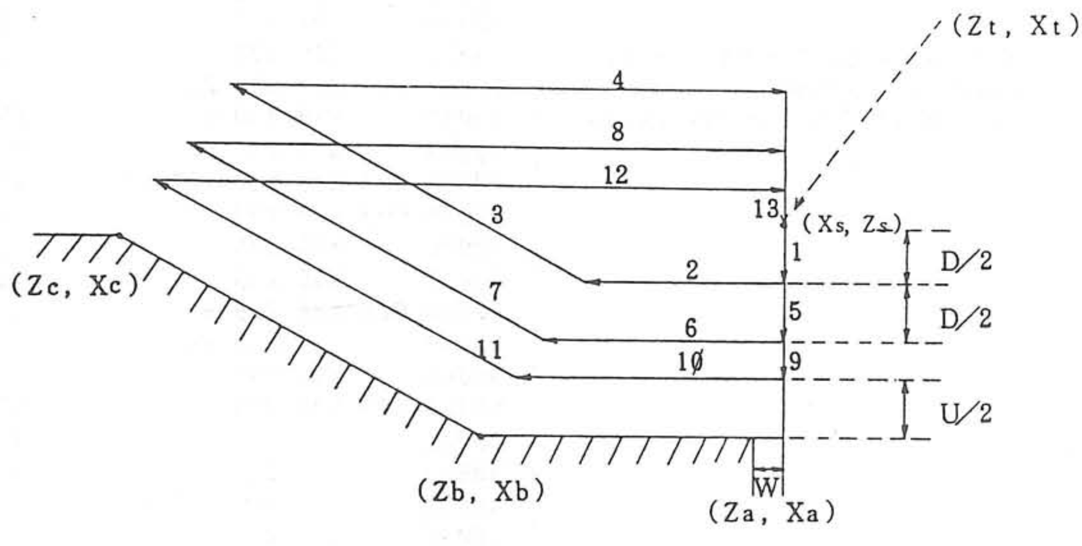


Fig. 1Ø-2 Tool Paths in Copy Turning Cycle

1Ø-2-2. Tool Nose R Compensation

(1) G Codes

G41, G42	Nose R compensation command
G4Ø	Nose R compensation command cancel

(2) Example of Conversion of Nose R Compensation Commands

Tool paths generated by the use of tool nose R compensation commands (Fig. 1Ø-3) are converted into a general program.

<Input Program>

```
NØØØ1 GØØ XØ ZØ
NØØØ2 GØ1 X1 Z1
NØØØ3     X2 Z2 G42 F1 TØ1Ø1Ø1
NØØØ4     X3 Z3
NØØØ5     X4 Z4
NØØØ6     X5 Z5
NØØØ7 G4Ø
NØØØ8 GØØ X6 Z6

          :
          :
```

<Output Program>

```
NØØØ1 GØØ XØ' ZØ'
NØØØ2 GØ1 X1' Z1'
NØØØ3     X2' Z2' F1 TØ1Ø1
NØØØ4     X3' Z3'
NØØØ5     X4' Z4'
NØØØ6     X5' Z5'
NØØØ7 GØØ X6' Z6'
          :
          :
```

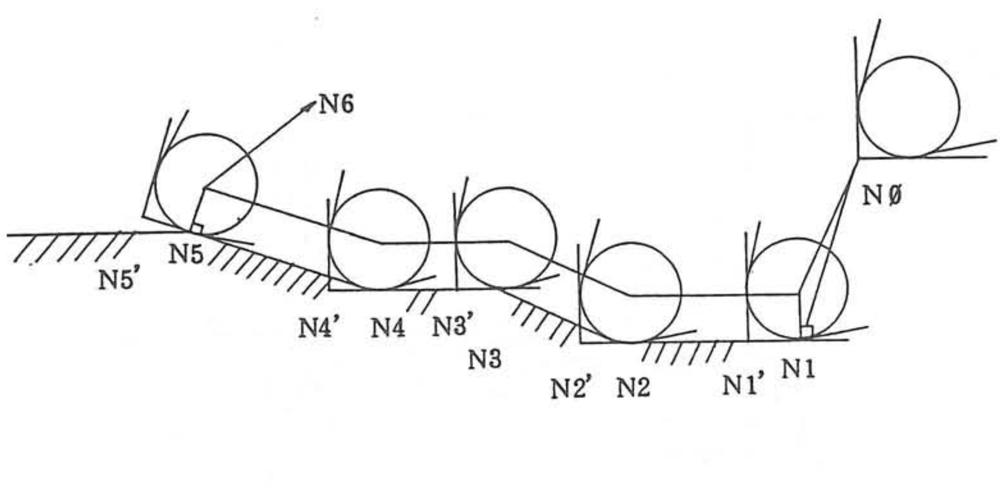


Fig. 1Ø-3 Tool Paths with Tool Nose R Compensation Command Active

1Ø-2-3. Compound Fixed Cycle

(1) G Codes and M Codes

G71: Longitudinal thread cutting compound fixed cycle
G72: Transverse thread cutting compound fixed cycle*
G73: Longitudinal grooving fixed cycle
G74: Transverse grooving fixed cycle
M32: Straight infeed along thread face (on left face)*
M33: Zigzag infeed*
M73: Infeed pattern 1
M74: Infeed pattern 2
M75: Infeed pattern 3

* Only for OSP5ØØØ

(2) Example of Conversion of G71 (longitudinal thread cutting compound fixed cycle)

a) Conversion for OSP5ØØØ

The G71 cycle is converted into the G33 (longitudinal thread cutting cycle)

<Input Program>

```
NØØØ1 GØØ Xs Zs
NØØØ2 G71 Xa Za Ii Bb  Dd  Uu  Hh L1
  $          Ff Jj M23 M32 M75

        :
        :
```

<Output Program>

```
NØØØ1 GØØ Xs  Zs
NØØØ2 G33 Xa1 Za Ii K1 L1 Ff Jj M23
NØØØ3     Xa2       K2
NØØØ4     Xa3       K3
NØØØ5     Xa4       K4

           :
           :
```

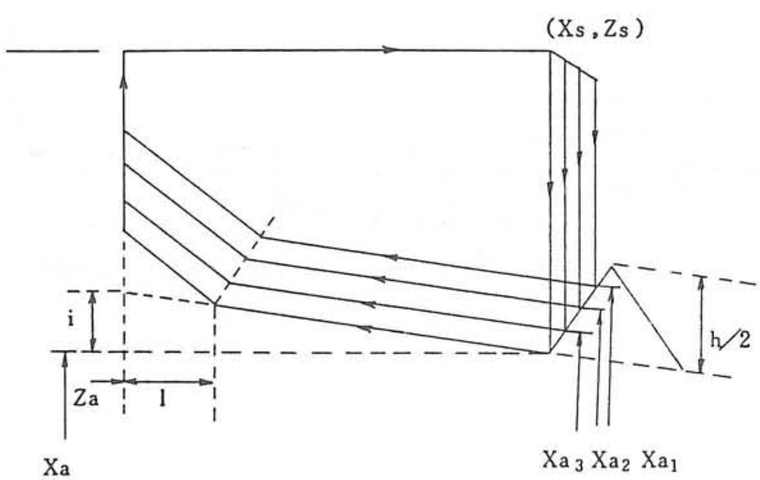


Fig. 1Ø-4 Tool Paths in Longitudinal Thread Cutting Compound Fixed Cycle (for OSP5ØØØ)

b) Conversion for OSP3ØØØL

The G71 cycle is converted into the G33 (longitudinal thread cutting cycle).

<Input Program>

```
NØØØ1 GØØ Xs Zs
NØØØ2 G71 Xa Za Ii Dd Uu Hh Ff M75
      :
      :
```

<Output Program>

```
NØØØ1 GØØ Xs  Zs
NØØØ2 G33 Xa1 Za Ii Ff
NØØØ3     Xa2
NØØØ4     Xa3
NØØØ5     Xa4
           :
           :
```

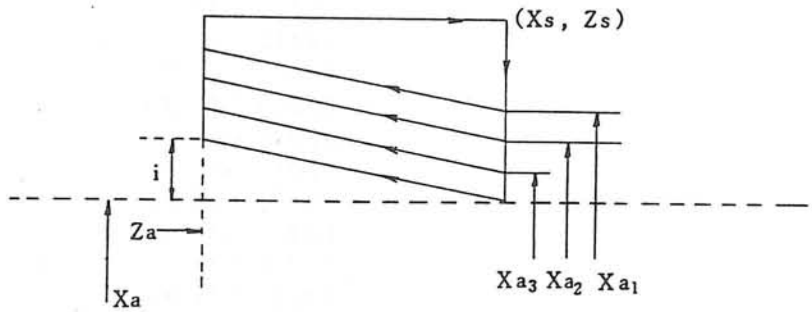


Fig. 1Ø-5 Tool Paths in Longitudinal Thread Cutting Compound Fixed Cycle (for OSP3ØØØL)

Note 1: The thread cutting compound cycle cannot be converted into the program for the OSP3ØØØL in the following cases.

a) B command (tool point angle) is specified in G71 (longitudinal thread cutting compound fixed cycle). Conversion is possible if B=Ø.

b) Both M23 (chamfering) and L (chamfering amount) are specified in G71.

c) G72 (transverse thread cutting compound fixed cycle) cannot be converted.

Note 2: If the J command (no. of threads) is specified, F commands are converted into a value F/J.

(2) Example of Conversion of G73 (longitudinal grooving fixed cycle)

The tool paths generated in the G73 cycle (Fig. 1Ø-6) are converted into a general program.

<Input Program>

```
NØØØ1 GØØ Xt Zt Ss TØ1Ø1 MØ3 M42
NØØØ2     Xs Za
NØØØ3 G73 Xe Ze Ii Kk Dd Ee Ff TØ1Ø2

      :
      :
```

<Output Program>

```
NØØØ1 GØØ Xt  Zt Ss TØ1Ø1 MØ3 M42
NØØØ2     Xs  Zs
NØØØ3     Xa
NØØØ4 GØ1 Xb     Ff
NØØØ5 GØØ Xb'
NØØØ6 GØ1 Xc
NØØØ7 GØØ Xc'
NØØØ8 GØ1 Xe
NØØØ9 GØ4        Fe
NØØ1Ø GØØ Xs
NØØ11         Za
NØØ12     Xa
NØØ13 GØ1 Xb
NØØ14 GØØ Xb'
NØØ15 GØ1 Xc
NØØ16 GØØ Xc'
NØØ17 GØ1 Xe
NØØ18 GØ4        Fe
NØØ19 GØØ Xs
NØØ2Ø         Zb
NØØ21     Xa
NØØ22 GØ1 Xb
NØØ23 GØØ Xb'
NØØ24 GØ1 Xc
NØØ25 GØØ Xc'
NØØ26 GØ1 Xe
NØØ27 GØ4        Fe
NØØ28 GØØ Xs

          :
          :
```

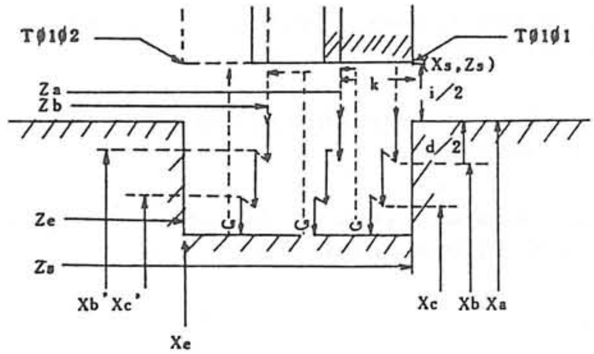


Fig. 1Ø-6 Tool Paths in Longitudinal Grooving Fixed Cycle

1Ø-2-4. Other Conversion Items

(1) The constant speed cutting commands G11Ø (for A-turret) and G111 (for B-turret) are ignored when conversion is made for the OSP3ØØØL.

(2) Program name "Oxxxx" is ignored when conversion is made for the OSP3ØØØL.

1Ø-2-5. Unit System for Conversion

The unit system used for the IGF conversion follows the setting of the optional parameter (bit) No. 3. For details, refer to the Operation Manual (Publication No. 2446-E).

1Ø-2-6. Tool Data Used in Conversion

The tool data, such as tool offset data and tool nose radius compensation data, set in the TOOL DATA setting process from the NC is used for the conversion. For details, refer to the Operation Manual (Publication No. 2446-E).

Interactive Graphic MDI Function

IGF-L3

OSP500L-G/OSP5000L-G CNC SYSTEMS

OPERATION MANUAL —APPLICATION—

(4th Edition)

OKUMA

Your single source for machine & control

Publication No. CB002 2476-E-R3
(April 1987)

TABLE OF CONTENTS

PAGE

PAGE

Note: To avoid any confusion over the use of the letter "O (oh)" and figure "0 (zero)" in this manual, the numerical value "0 (zero)" is expressed as "Ø" if there is any possibility of misunderstanding.

SECTION 1 IGF TEMPORARY QUIT AND RECOVERY FUNCTION

There are cases in which IGF operations are required to be temporarily suspended during graphic edit operation to make it possible to carry out some other necessary operations (to change to a different operation mode) or when graphic edit operations cannot be completed up to the end of the input data due to insufficient available bubble memory space.

In such cases, it is of course possible to quit the IGF mode at the graphic edit operation currently executed. However, this requires several steps and not effective when urgent quit is required.

The IGF temporary quit and recovery function allows the graphic edit operation, interrupted by mode change, etc. without quitting the IGF operation to be recovered from the point where the IGF operations were left off.

Quitting Method:

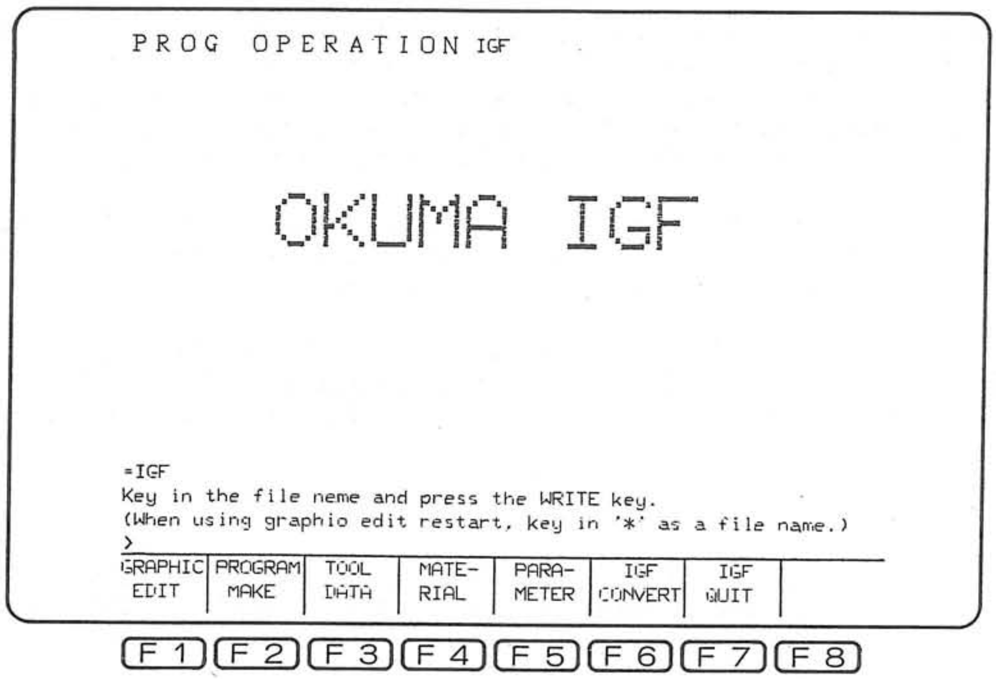
Press the mode selection switch (EDIT AUX., PARAMETER, ZERO SET, TOOL DATA, AUTO, MDI and MANUAL) during the IGF operation. This interrupts the graphic edit operation.

Recovery Method:

1) After the completion of required operation in other operation mode, select the EDIT AUX. mode first and then press function key [F7] (IGF).

2) Press function key [F1] (GRAPHIC EDIT) and key in "*" instead of a file name, then press the WRITE key.

This allows the IGF operations to be continued with the file name for which graphic edit was being carried out before the temporary quit. In this file all processes created before the temporary quit are saved.



Precautions:

(1) If the IGF is interrupted in the process of shape designation, the process data of the preceding process is saved. The shape being designated is lost.

(2) Recovery of the graphic edit operation is possible until control power is turned off.

(3) If the following operation is conducted in the IGF operation before the recovery of temporarily quit IGF, recovery is impossible:

Program creation (F2)
Setting of tool data (F3), material data (F4),
parameter (F5)

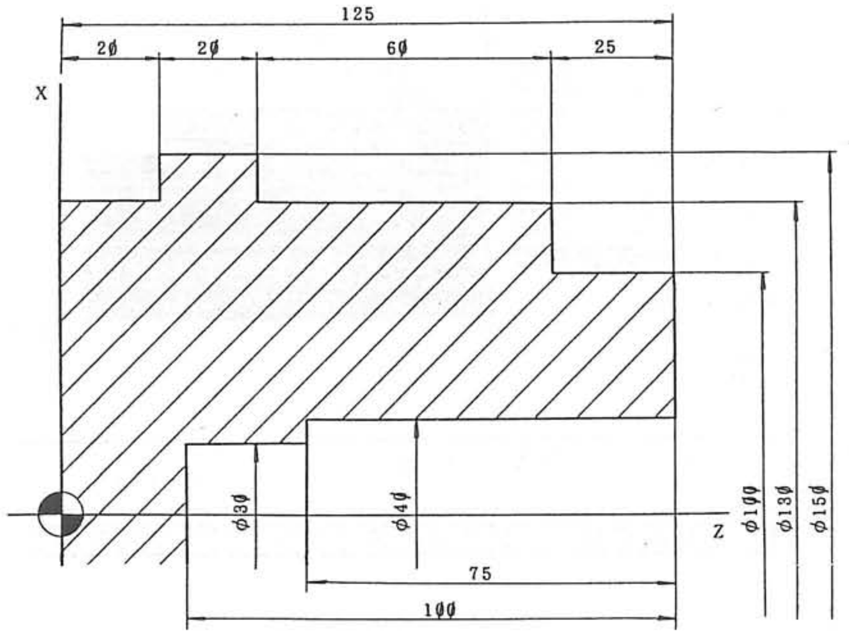
Note that the IGF CONVERT (F6 option) operation does not influence the recovery of the IGF operation.

(4) The recovery function is not influenced by the edit or transfer operation in the program operation mode.

SECTION 2 ENTRY OF MULTI-DIAMETER BLANK SHAPE

In the BLANK SIZE definition process of the program edit function, blank dimensions with up to six outside diameters (OD) and four inside diameters (ID) can be defined.

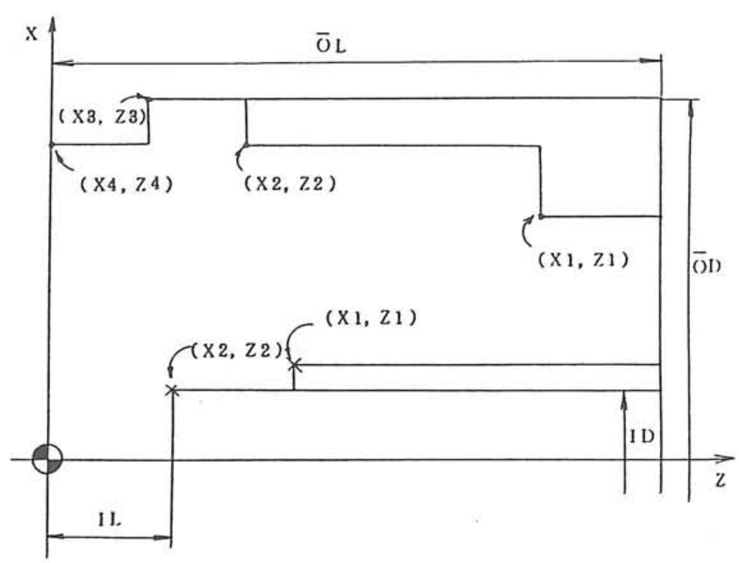
This section provides an example for defining the blank illustrated below:



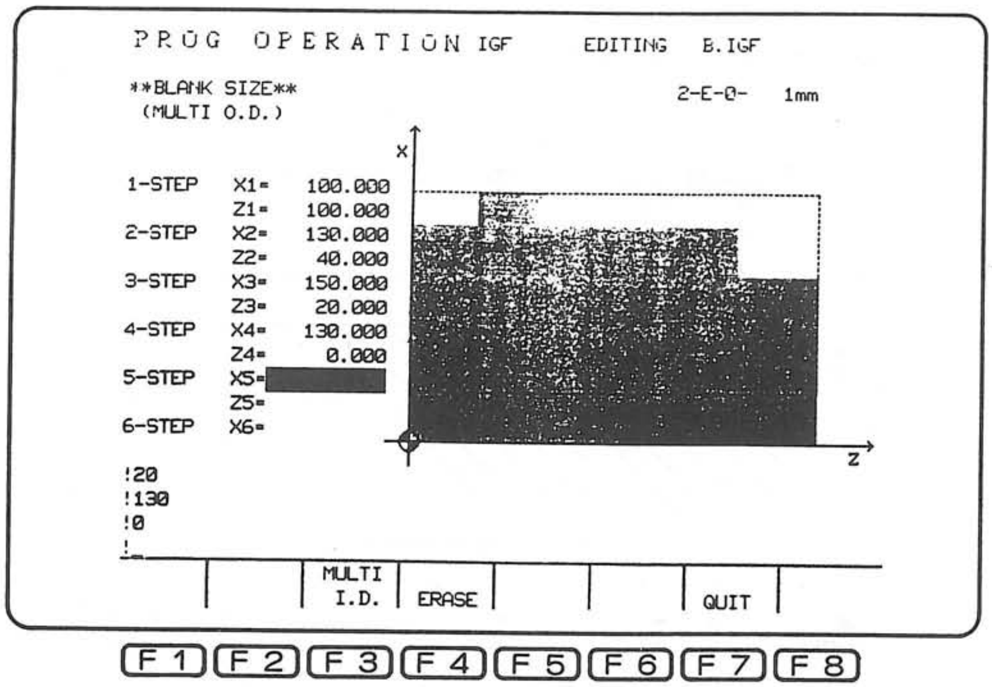
Procedure:

1) From the BLANK SIZE (BASIC SIZE) page, enter the dimensions defining the blank envelope.

OUT DIA	OD=15Ø.ØØØ
OUT LENG	OL=125.ØØØ
IN DIA	ID= 3Ø.ØØØ
IN LENG	IL= 25.ØØØ
DZ SHIFT	ZS= Ø.ØØØ
PZ SHIFT	ZO= Ø.ØØØ



2) After the completion of data entry as indicated above, press the function key [F3] (MULTI O.D.). The display should change to the BLANK SIZE (MULTI O.D.) page.

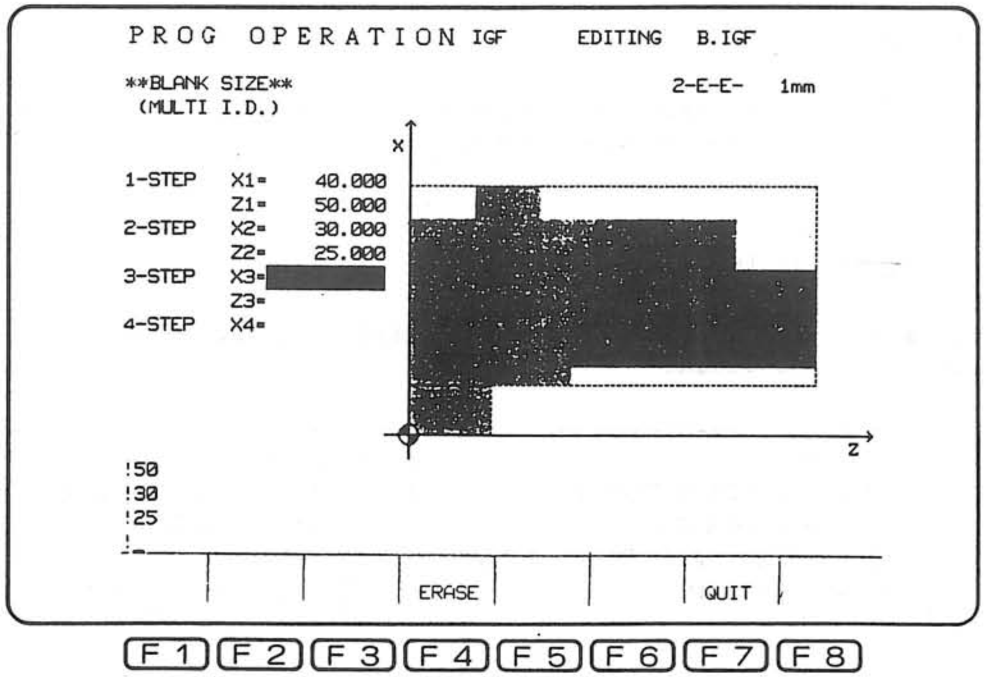


Enter the data as indicated above.

For changing the data entered, locate the cursor on the data and key in the new data.

3) Press the function key [F3] (MULTI I.D.).

Enter the data as indicated below.



Note that if the blank ID is zero, data entry is not allowed.

For changing the data entered, locate the cursor on the data and key in the new data.

4) Press the function key [F7] (QUIT).

This completes the entry of multi-diameter blank size.

SECTION 3 PROCESS EDITING

Data entered can be edited in units of processes. This editing operation includes insertion, copy and deletion.

3-1. INSERTION

This shifts the processes following the specified process downward by one, leaving the specified process blank.

Example:

[INSERT] [2] [WRITE]

This shifts the processes from PROCESS NO 2 backward and PROCESS NO 2 area is left blank.

```
PROCESS                              PROCESS
NO 1=A ROUGH O.FACE ↓                NO 1=A ROUGH O.FACE ↓
NO 2=A ROUGH OD     ←                NO 2=  [blank]
NO 3=A FIN. O.FACE  ↓       ⇒        NO 3=A ROUGH OD     ←
NO 4=A FIN. OD      ←                NO 4=A FIN. O.FACE  ↓
   :       :                         NO 5=A FIN. OD      ←
   :       :                            :       :
```

3-2. COPY

This function copies the process data of the specified process to the blank process. If more than one blank process are present, copy is made to the first blank process. Copying the process to the specified process is also possible.

Example 1:

[COPY] [3] [WRITE]

This copies the process data of PROCESS NO 3 to the first blank process.

```
PROCESS                              PROCESS
NO 1=A ROUGH O.FACE ↓                NO 1=A ROUGH O.FACE ↓
NO 2=  [blank]                       NO 2=A FIN. O.FACE  ↓
NO 3=A FIN. O.FACE  ↓                NO 3=A FIN. O.FACE  ↓
NO 4=A FIN. OD      ←                NO 4=A FIN. OD      ←
NO 5=A GROOVE OD    ↓                NO 5=A GROOVE OD    ↓
NO 6=A THREAD OD    ←                NO 6=A THREAD OD    ←
NO 7=  [blank]                       NO 7=  [blank]
   :       :                            :       :
```

Example 2:

[COPY] [2] [,] [5] [WRITE]

This copies the process data of PROCESS NO 2 to the specified blank process PROCESS NO 5.

```
PROCESS
NO 1=A ROUGH O.FACE ↓
NO 2=A ROUGH OD     ←
NO 3= blank
NO 4=A FIN. OD      ←
NO 5= blank
  :       :
```

⇨

```
PROCESS
NO 1=A ROUGH O.FACE ↓
NO 2=A ROUGH OD     ←
NO 3= blank
NO 4=A FIN. OD      ←
NO 5=A ROUGH OD     ←
  :       :
```

3-3. DELETION

The process specified is deleted and the processes that follow are shifted upward to fill the deleted process.

Example:

[DELETE] [1] [WRITE]

```
PROCESS
NO 1=A ROUGH O.FACE ↓
NO 2=A ROUGH OD     ←
NO 3=A FIN. O.FACE  ↓
  :       :
```

⇨

```
PROCESS
NO 1=A ROUGH OD     ←
NO 2=A FIN. O.FACE ↓
  :       :
  :       :
```

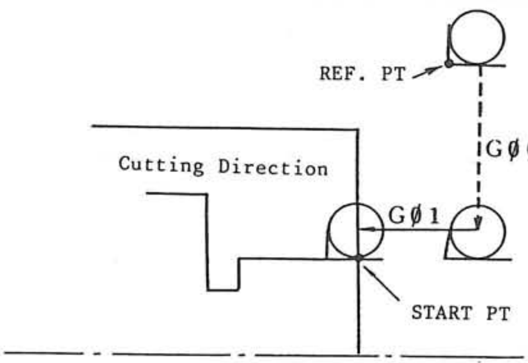
SECTION 4 PROCESS KINDS

4-1. ROUGH, COPY AND FINISH

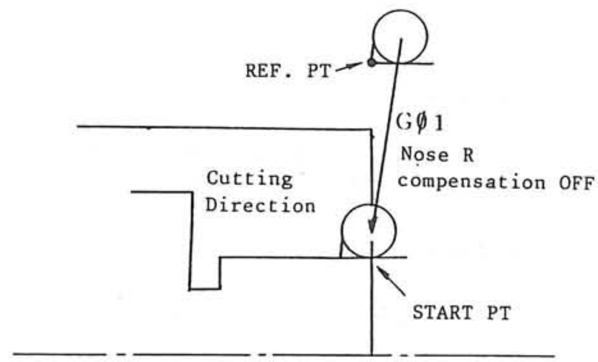
(1) Tool Movements from Reference Point (REF. PT) to Start Point (START PT)

a) For REF. PT and START PT, enter different values for both X and Z to assure safety of operations.

Example 1: Both X and Z different for REF. PT and START PT (OD ROUGH)



Example 2: For REF. PT and START PT, Same Z Data Entered (OD ROUGH)



b) When both X and Z data of parameters REF. PT and START PT are different, positioning is first carried out in the direction right angles to the specified cutting direction at a rapid traverse speed and, then, positioning at the START PT is carried out at a cutting feedrate. In this second positioning movement, the tool nose R compensation function is activated. See Example 1.

c) If either of X or Z data of REF. PT and START PT is identical, positioning from the REF. PT to the START PT is carried out directly at a cutting feedrate. The tool nose R compensation function is activated in this positioning cycle. See Example 2.

d) Note that in the MID cutting pattern, the positioning from the REF. PT to the START PT is always carried out directly. Feedrates used in this positioning depend on the process kind being selected. That is, in the ROUGH and FINISH processes, the positioning is carried out at a cutting feedrate and in the COPY process, the positioning is carried out in a rapid traverse.

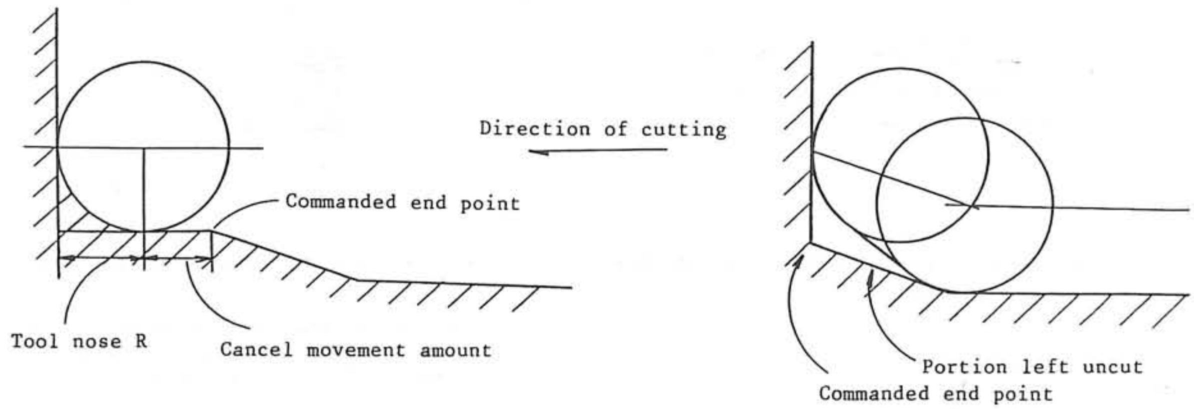
(2) Tool Movements at End Point of Shape Definition

a) At the end of shape definition, the cutting tool is always fed in the same direction as the cutting direction by the amount equivalent to the tool nose R compensation amount.

This tool nose R compensation cancel movement can eliminate portion uncut as shown in Example 2 below.

Example 1: Cancel movement amount ≒ Ø

Example 2: Cancel movement amount = Ø



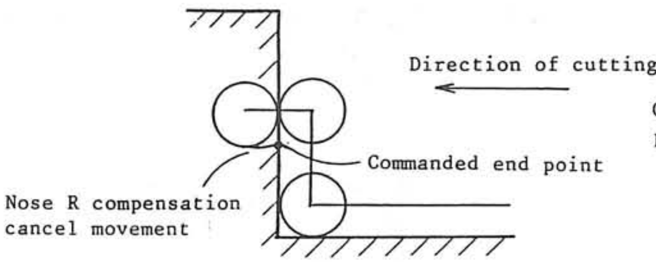
b) The tool nose R compensation cancel movement amount is set by the IGF dimension parameter.

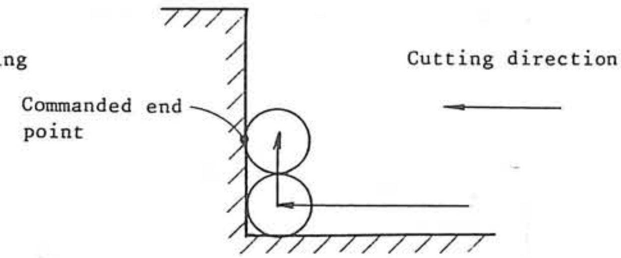
c) By this cancel movement, the cutting tool is fed beyond the commanded end point by the amount calculated in the following formula.

Nose R + Nose R compensation cancel amount

d) The tool nose R cancel movement causes overcutting if cutting is ended at a mid point on a workpiece. To eliminate this overcutting, set "Ø" for the tool nose R cancel movement amount. See the examples below.

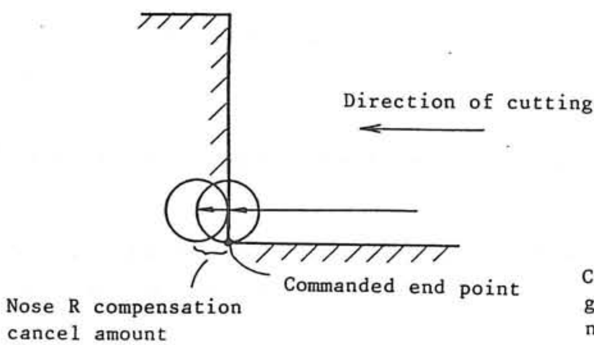
Example 1: Cancel movement amount ≒ Ø ——→ Cancel movement amount = Ø

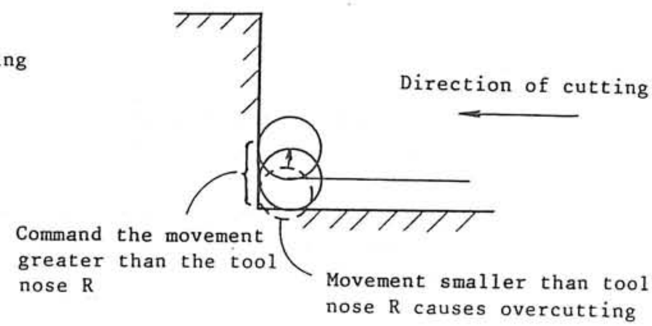




Example 2: Cancel movement amount ≒ Ø ——→ Cancel movement amount = Ø

(Define a shape in the X-axis direction which requires movement larger than tool nose R, with cancel amount "Ø")





4-2. THREAD CUTTING

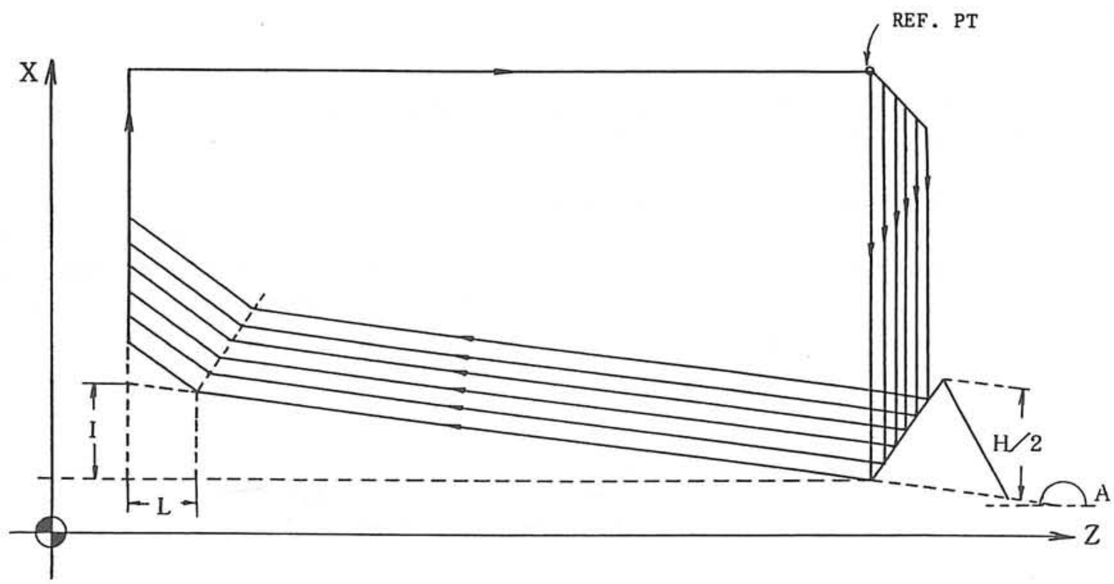
(1) Longitudinal Thread Cutting

Parameters used for defining thread cutting process are indicated below:

REF. PT	X=	Reference point X (in diameter)
	Z=	Reference point Z
START PT	XA=	Start point X (root diameter or nominal diameter)
END PT	XB=	End point X (root diameter or nominal diameter)
	ZB=	End point Z
HEIGHT	H=	Thread height (in diameter)
TAPER	I=	Slope of taper thread*
Z ANGLE	L=	Angle of taper thread**

* Effective when XB is not specified
** Effective when XB and I are not specified

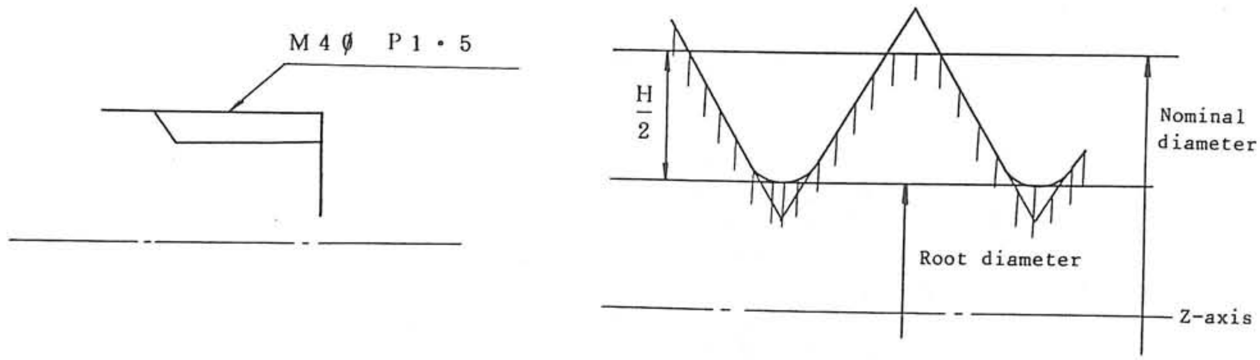
The longitudinal thread cutting definition required at least the parameters XA, ZB, H and any of XB, I or A.



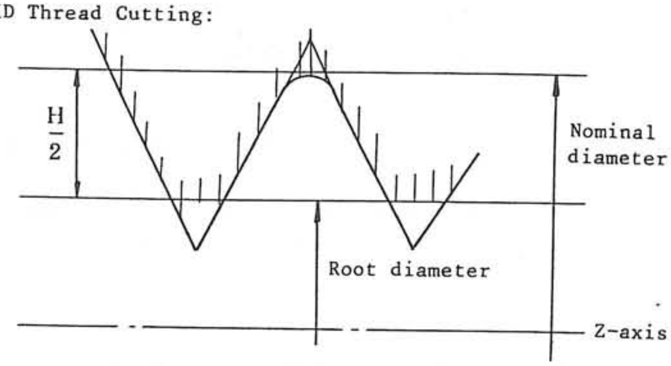
For parameters START PT XA and END PT XB, either the root diameter or the thread nominal size can be used. The IGF integer parameter sets which of these data type should be used. Standard setting is root diameter.

The parameters REF. PT Z and END PT ZB should be designated taking leading and ending incomplete thread portion into consideration.

Thread Height and Thread Diameter:



ID Thread Cutting:



The thread height (H) is preset at the values calculated in the following formulae.

OD threading	H = F/(J x N) x $\boxed{65\%}$	
ID threading	H = F/(J x N) x $\boxed{50\%}$	← These values are set at IGF percent parameter and changeable as required.
Face threading	H = F/(J x N) x $\boxed{87\%}$	

where,

F = lead
J = number of threads
N = number of thread leads

For thread cutting, thread cutting mode and thread cutting pattern can be designated by the IGF integer parameters. For details, refer to the Instruction Manual for Optional Specifications.

(2) Thread Cutting on Face

The parameters used for defining the thread cutting process are identical to those used for defining longitudinal thread cutting process.

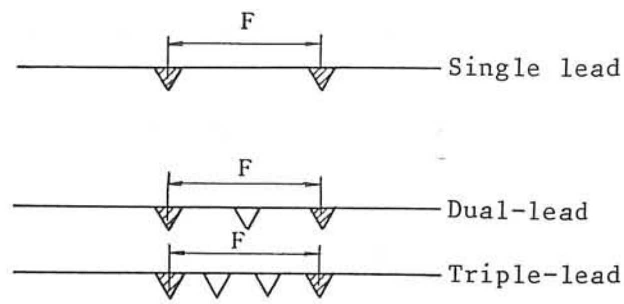
(3) Multiple Lead Thread Cutting

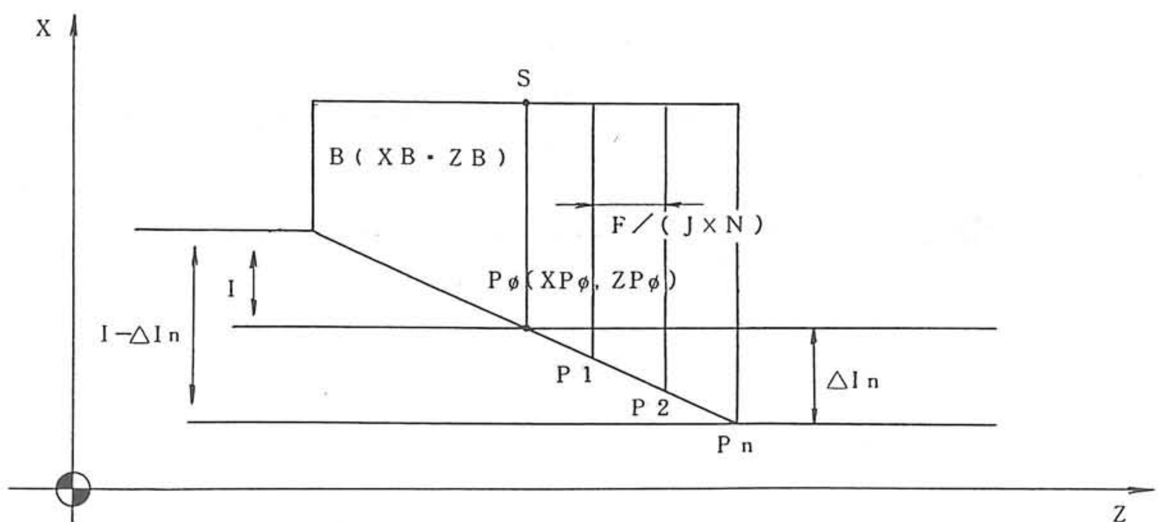
The number of thread leads can be specified by cutting condition A. For the parameter designating the multiple lead thread cutting, default which is set by the IGF parameter is preset.

Cutting conditions:

```
**PROCESS**
  (CUT CONDITION)

  LEAD        F =
  TPI         J = 1
  LEAD NO.    NT= 1
```

Example:





$$\triangle In = \frac{1}{ZB - ZP\phi} \times F / (J \times N) \times n$$

(4) Tapping

The data to be input for tapping operation are as indicated below:

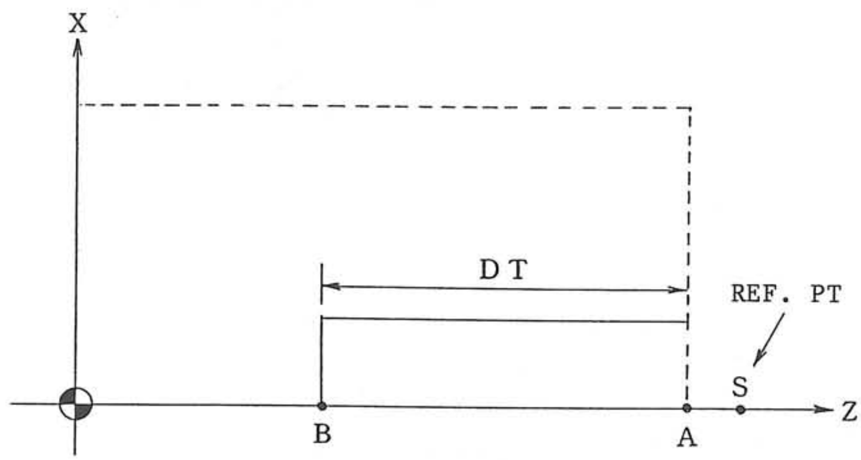
REF. PT Z= Reference point

START PT ZA= Tapping starting point

END PT ZB= Tapping end point

DEPTH DT= Distance from the starting point (depth)

This step ends after the input of ZA data and either of ZB or DT data.



4-3. GROOVING

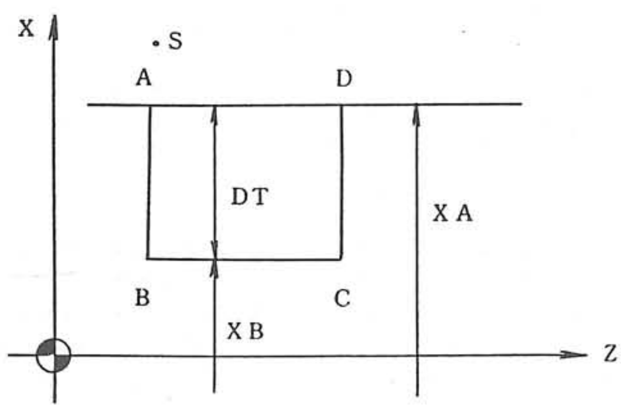
(1) Longitudinal Grooving

Parameters used for defining grooving process are indicated below:

REF. PT	X= Reference point X (in diameter)
	Z= Reference point Z
START PT	XA= Start point X (in diameter)
END PT	XB= End point X (in diameter)
DEPTH	DT= Groove depth (in radius)*
GROOVES	NG= Number of grooves
SHIFT	K= Shift amount
POINT A	AK= Chamfer type at point A**
CHF SIZE	AS= Chamfer size
POINT D	DK= Chamfer type at point D**
CHF SIZE	DS= Chamfer size

* Effective when XB is not specified
** (Ø: C-CHF 1: R-CHF)

The longitudinal grooving definition requires at least the parameters XA and either XB or DT.



a) Parameter Data Entry

Groove position is determined based on REF. PT Z (for longitudinal grooving) or REF. PT X (for face grooving). Tool offset values are active when positioning at the groove cutting point is carried out.

Cutting starts from point S.

Depth DT should always be commanded in a positive value disregarding of the direction of cuts.

When pecking is not specified, step-infeed and step-retraction amounts are assigned two times the distance between point S and point B on an NC program compiled from the IGF data. Thus, actual cutting operation does not include the pecking cycle.

The combination of parameters NG and K is used for specifying multiple grooving of the identical shape groove. Default of the parameter NG is as set by the IGF integer parameter. Multiple grooving is called for when parameter K=Ø and setting of parameter NG is greater than one (NG > 1).

Parameters AK and DK specify the type of chamfering. The selection is made from menu - Ø: C-CHF; 1: R-CHF.

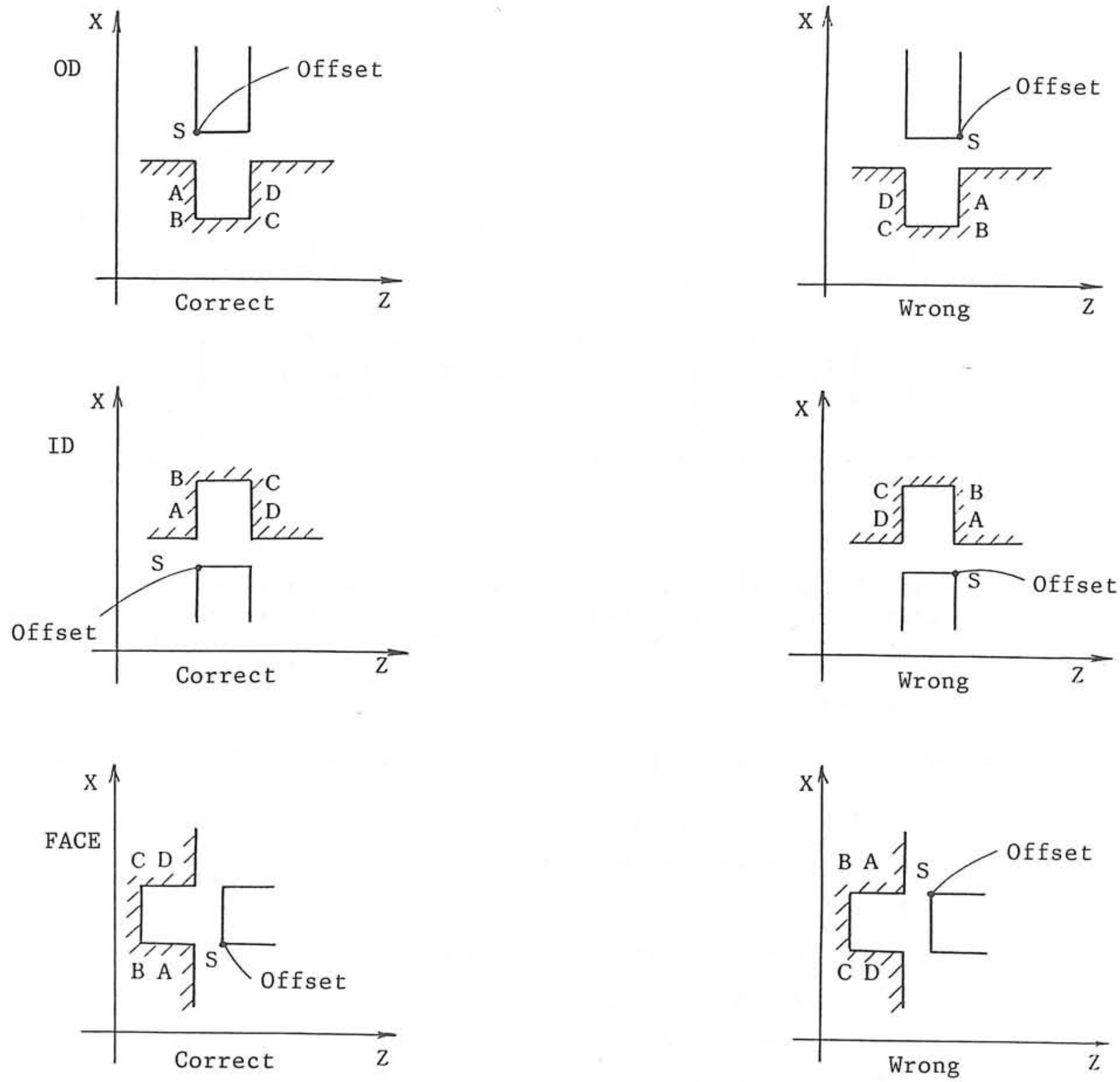
Parameters AS and DS specify the size of chamfering.

b) Tool Offset in Positioning at Reference Point

Set the tool nose radius of the grooving tool at parameter NOSE RADIUS R= on the TOOL DATA setting page.

Nose radius set in this step is not reflected to the data file registered in the NC.

Offset values used for positioning at the reference point for groove cut should be determined in reference to the figures below depending on OD, ID and face grooving operations. Positioning of the grooving tool is carried out with the indicated edge taken as the reference and groove position is determined by this positioning.

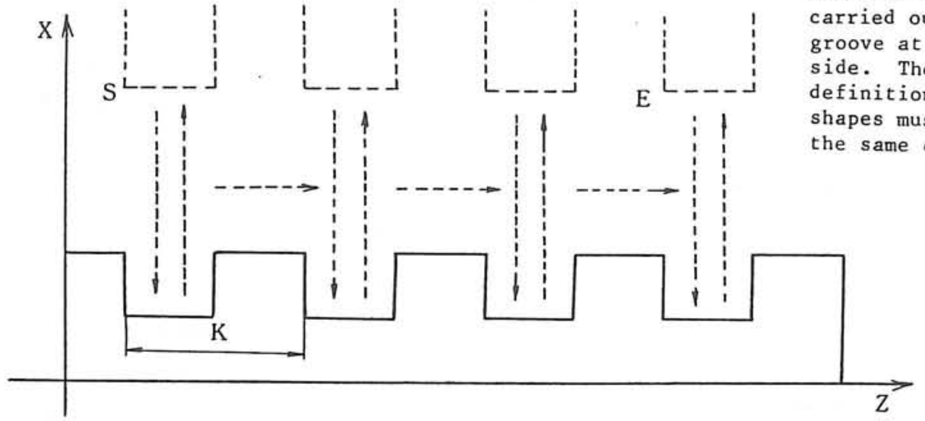


That is, when point A at the positioning side is taken as (XA, ZA) and point D at the opposite side of the positioning point is taken as (XD, ZD), then the following inequalities should be met:

$$\left(\begin{array}{l} \text{For OD/ID grooving ...... } ZA < ZD \\ \text{For Face grooving .... } XA < XD \end{array}\right)$$

If these inequalities are not met, the shape machined cannot be accurate.

c) Positioning for each Grooving Cycle in Multiple-Groove Machining Cycle



When multiple grooves are machined, it is carried out from the groove at the chuck side. Therefore, definition of groove shapes must be made in the same order.

After the completion of grooving of multiple grooves, the tool is returned to the cycle reference point X. In the case of face grooving, it is returned to the cycle reference point Z.

(2) Face Grooving

Parameters used for defining grooving process are indicated below:

REF. PT	X=	Reference point X (in diameter)
	Z=	Reference point Z
START PT	XA=	Start point X (in diameter)
END PT	XB=	End point X (in diameter)
DEPTH	DT=	Groove depth (in radius)*
GROOVES	NG=	Number of grooves
SHIFT	K=	Shift amount
POINT A	AK=	Chamfer type at point A**
CHF SIZE	AS=	Chamfer size
POINT D	DK=	Chamfer type at point D**
CHF SIZE	DS=	Chamfer size

* Effective when XB is not specified
** (Ø: C-CHF 1: R-CHF)

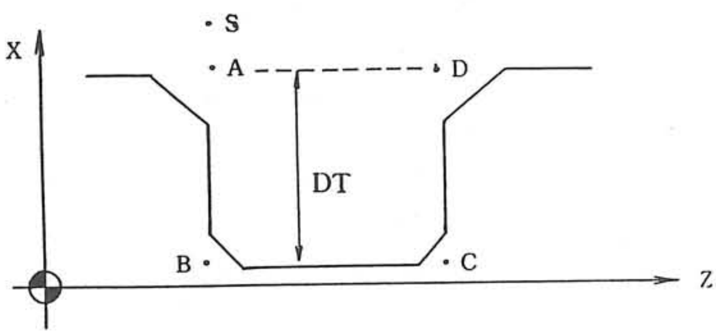
The face grooving definition requires at least the parameters XA and either XB or DT.

(3) Wide-Groove Machining

Parameters used for defining wide-groove process are indicated below:

```
START PT   XA= Start point
END PT     XB= End point
           ZD=
DEPTH      DT= Groove depth
POINT A    AK= Chamfer type at point A*
CHF SIZE   AS= Chamfer size
POINT A    BK= Chamfer type at point B*
CHF SIZE   BS= Chamfer size
POINT A    CK= Chamfer type at point C*
CHF SIZE   CS= Chamfer size
POINT D    DK= Chamfer type at point D*
CHF SIZE   DS= Chamfer size
           *  (Ø: C-CHF   1: R-CHF)
```

Chamfer type is determined by AK, BK, CK and DK, and chamfer size by AS, BS, CS, and DS.

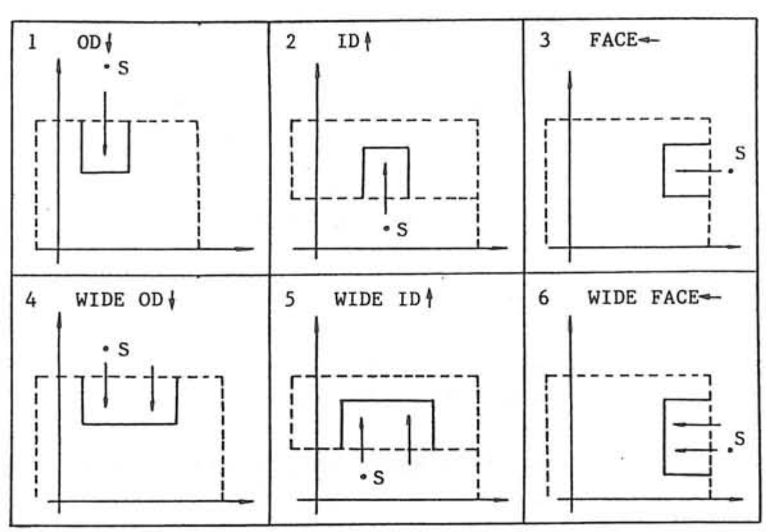


a) Direction of Cut

Cutting of a wide groove is carried out in the directions indicated below.

** PROCESS **
(CONDITION)
NO 3=GROOVE OD

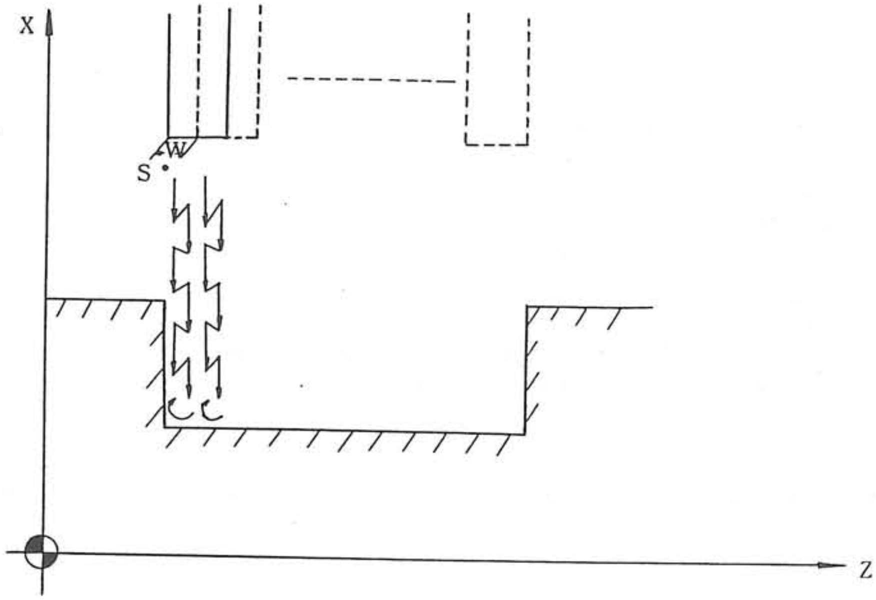
DIRECT ↓ NO=4



cutting direction: 1=OD ↓, 2=ID ↑, 3=FACE←, 4=WIDE OD ↓
5=WIDE ID ↑, 6=WIDE FACE←

b) Cutting Conditions

For "W", it is set in cutting condition A.



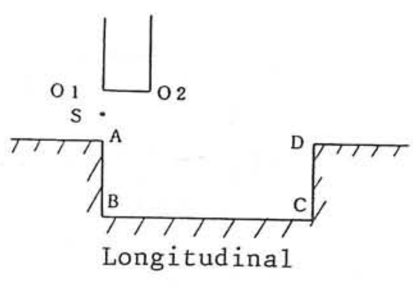
Cutting starts from point S, which should be set at the chuck side.

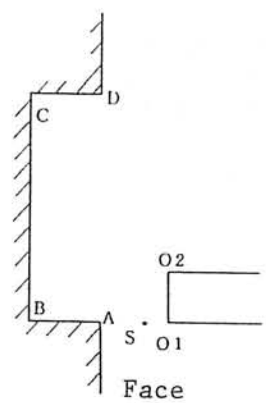
The shift width W has default of "tool width x P".

The tool width is set at tool data.

The parameter P which is used for determining the shift width can be changed by the IGF percent parameter. (Initial setting is 9Ø%.)

Tool offset O1 and O2:





O1: 1st offset
O2: 2nd offset

For "O1" and "O2", setting is possible at the tool designation process.

Positioning at the start point is carried out using the offset value at "O1". Concerning the tool nose radius, the tool nose radius set in the NC tool data file is used.

c) Finishing Conditions

Cutting condition B allows finish cutting. Note that the tool used for finishing should be the same one as used for roughing cycle. The cutting conditions, cutting speed and feedrate can be changed.

If the finishing allowance is "Ø", the defined shape is machined in the rough cutting mode.

(4) V-groove cutting

Parameters used for defining V-grooving process are indicated below:

START PT XA= Entrance point

ZA= Entrance point (This input is not necessary when KB or AB is specified.)

XB= Bottom point (This input is not necessary when DT is specified.)

KB= Bottom point (This input is not necessary when ZA or AB is specified.)

AB= Bottom point (This input is not necessary when ZA or KB is specified.)

ZC= Bottom point

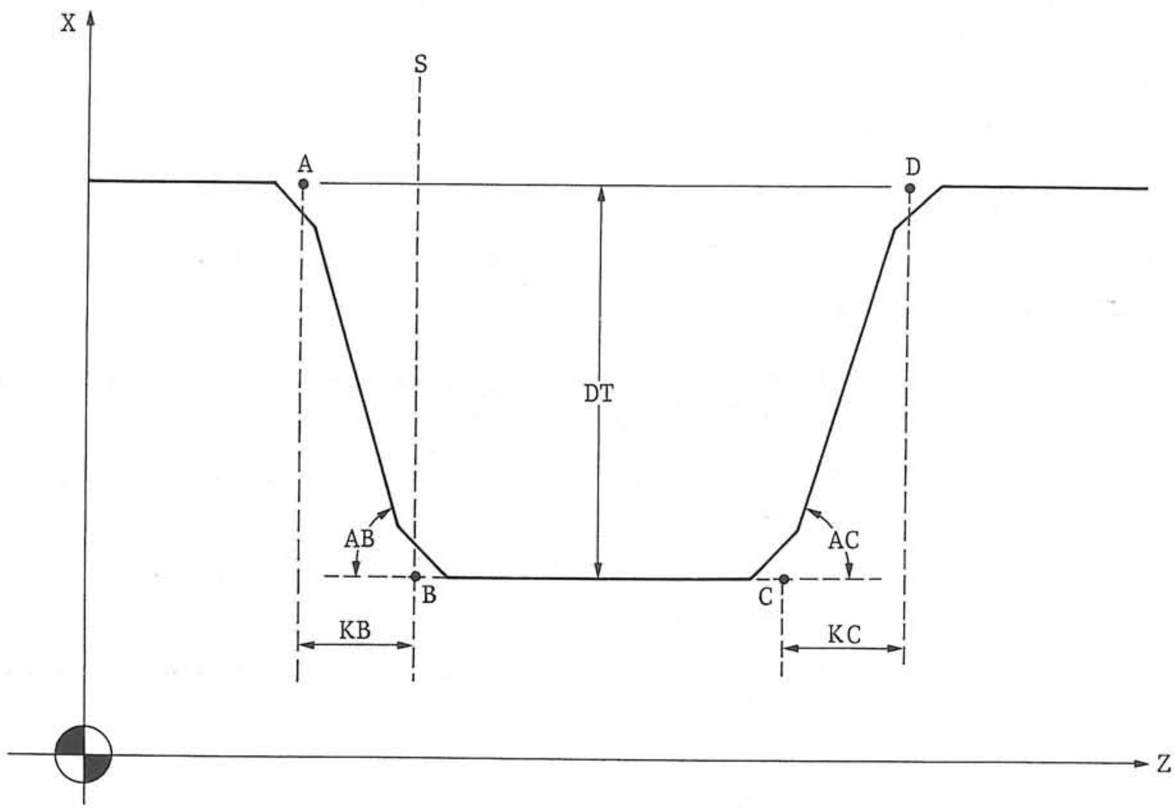
KC= Bottom point (This input is not necessary when ZD or AC is specified.)

AC= Bottom point (This input is not necessary when ZD or KC is specified.)

DT= Bottom point (This input is not necessary when XB is specified.)

AK to DK= Chamfer type at points A through D
(Ø: C-CHF 1: R-CHF)

AS to DS= Chamfer size


X
S
A
D
DT
AB
AC
B
C
KB
KC
Z

V-groove cutting is carried out in the following manner:

1) Rough cutting of the bottom

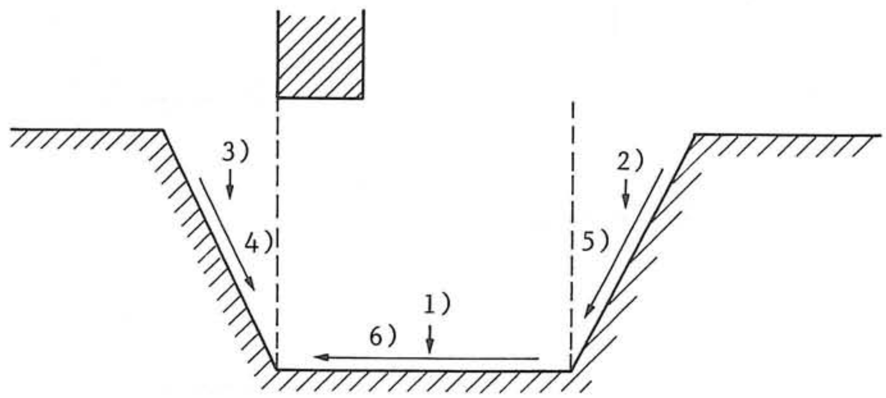
2) Rough cutting on the right hand side surface

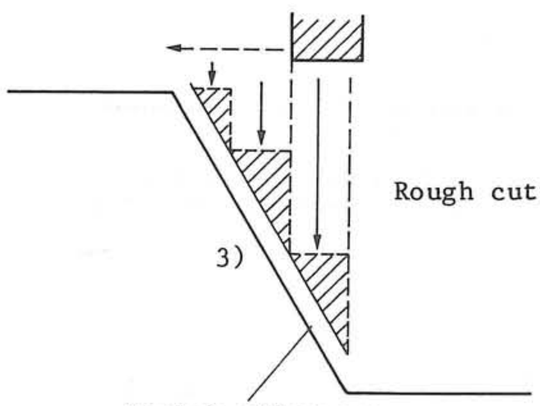
3) Rough cutting on the left hand side surface

4) Finish cutting on the left hand side surface

5) Finish cutting on the right hand side surface

6) Finish cutting on the bottom surface





Rough cutting on the side surface

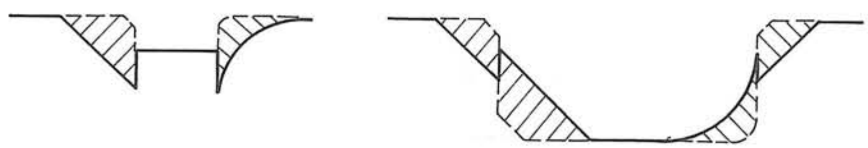
Finish allowance

Precautions for Groove Insertion Cycle:

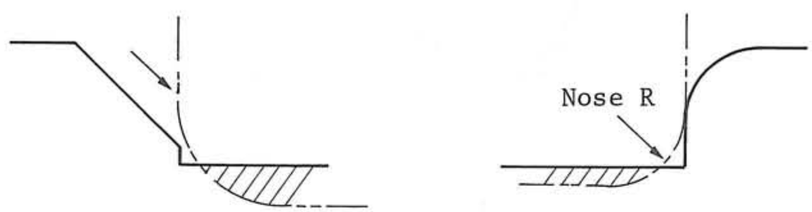
The IGF groove machining function has been designed with normal groove shapes in mind. This means that there will be occasions when factors such as chamfer size, nose R dimensions, groove width and the like make it impossible to cut a groove to a desired shape.

For this reason, the machine test and machine lock functions should always be used to provide an expanded display screen to check groove machining programs before they are executed.

a) Chamfer size and corner R too large

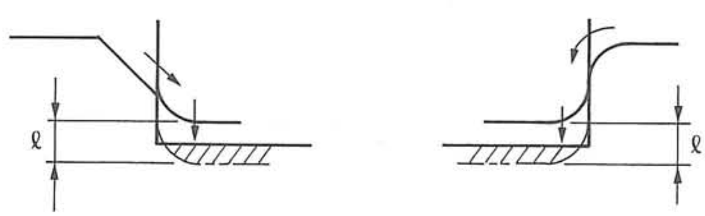


b) Nose R too large



c) Interference in groove cut due to deburring operation

After the chamfering operation in a groove cut deburring is conducted, and interference will occur during this operation.



ℓ: Movement by nose R amount from position where chamfering was competed.

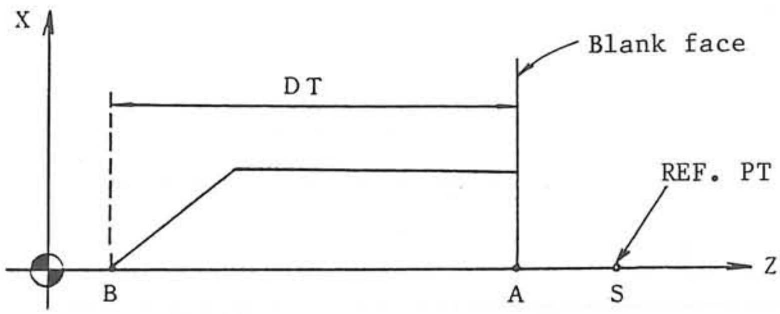
4-4. DRILLING

Parameters used for defining drilling process are indicated below:

REF. PT	Z=	Reference point Z
START PT	ZA=	Drilling cycle start point
END PT	ZB=	Drilling cycle end point
DEPTH	DT=	Drilling depth*

* Effective when ZB is not specified

The drilling definition requires at least the parameters ZA and either ZB or DT.



Drilling starts from point S. Drilling is usually carried out after centering operation. If the centering is not required, set "Ø" at parameter DEPTH D in the cutting condition table.

The parameter DEPTH DT always requires a positive value.

When pecking is not specified, step-infeed and step-retraction amounts are assigned two times the distance between point S and point B on an NC program compiled from the IGF data. Thus, actual cutting operation does not include the pecking cycle.

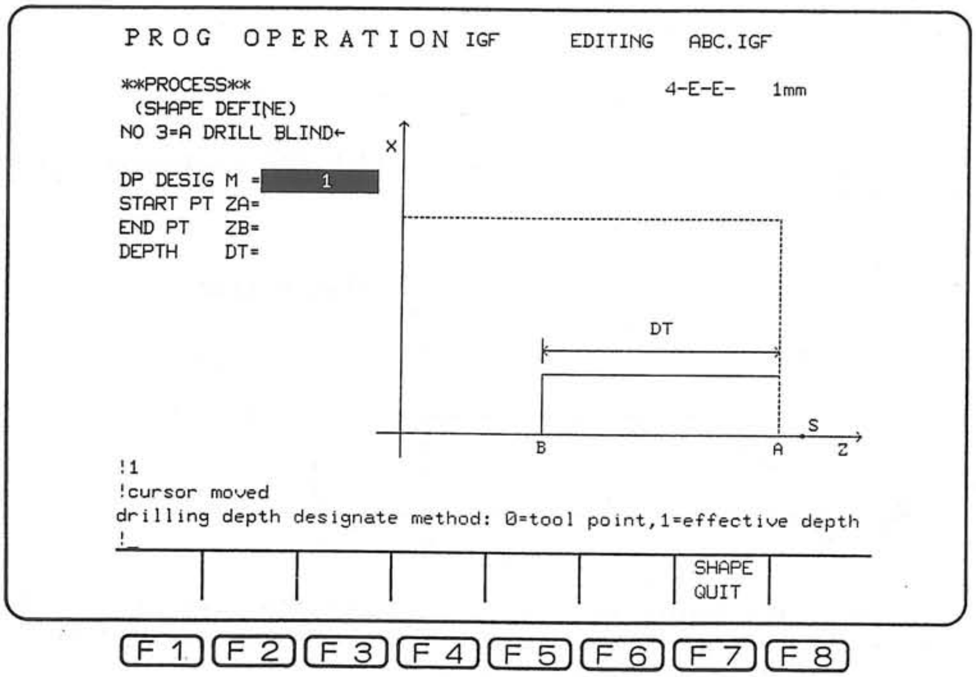
In the THRU drilling mode, drilling up to the point just before the workpiece is penetrated is carried out at the specified feedrate and the drill is retracted once. Then, the drill is again infed to the point near the first drilled depth at a rapid traverse rate. The drill is fed at the drilling-through feedrate specified by the IGF percent parameter. The IGF percent parameter for drilling through the workpiece is initially set at 5Ø%.

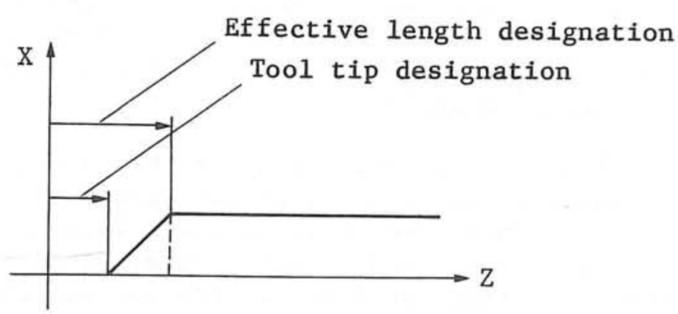
Dwell motion is not effective for centering operation.

After the completion of the drilling process, the ID clearance amount is automatically changed. For the ID clearance, refer to Section 1Ø-3, "Tool Change Control".

Drilling Effective Depth Designation

In order to specify effective depth for drilling, specify [1] for DP DESIG M= on the **PROCESS** (shape definition) page.





4-5. BALANCE CUT

With the 2-saddle specification, it is possible to cut machining time using the balance cut function. The processes to be machined in the balance cut mode are easily made through the IGF. The types of possible processes are shown below:

Rough cutting
Finish cutting
Copying
Groove cutting

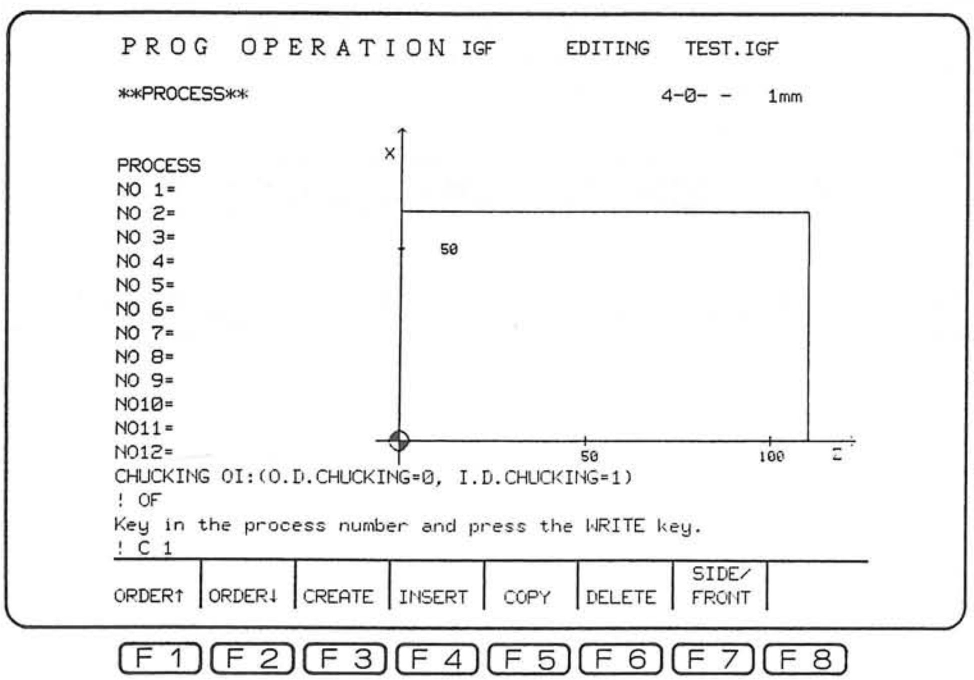
Cutting depth D and feedrate F of balance cut process can be changed by use of percent parameters (4Ø, 41).

When making a balance cut process:

Example:

1) Designate a process.

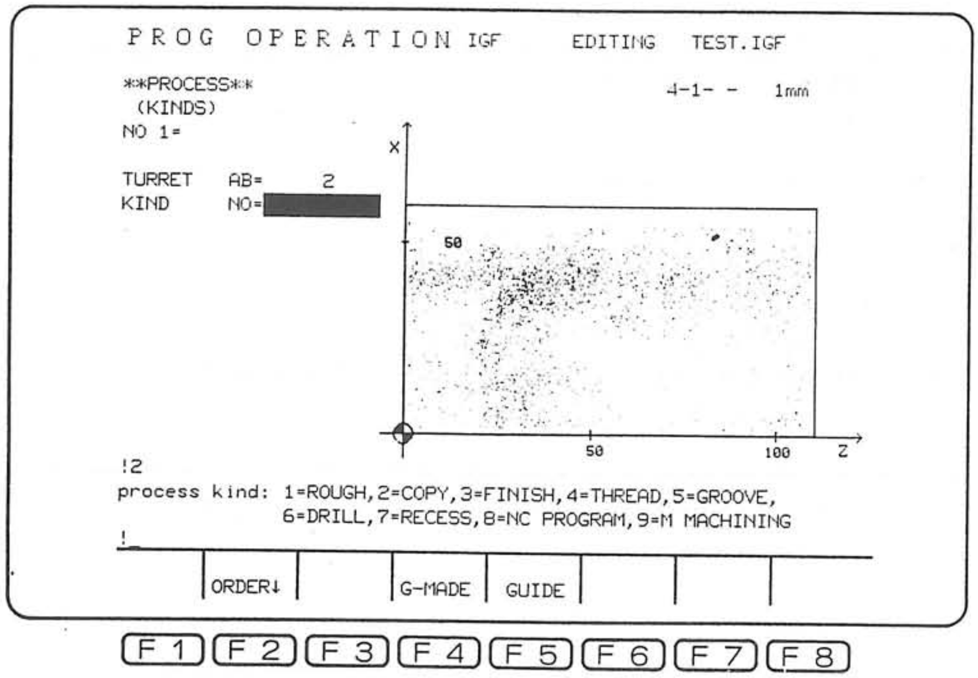
Here, the balance cut process is made in process 1.



2) Designate [2] when making a balance cut process.

The types of possible processes to be designated for balance cutting are shown below:

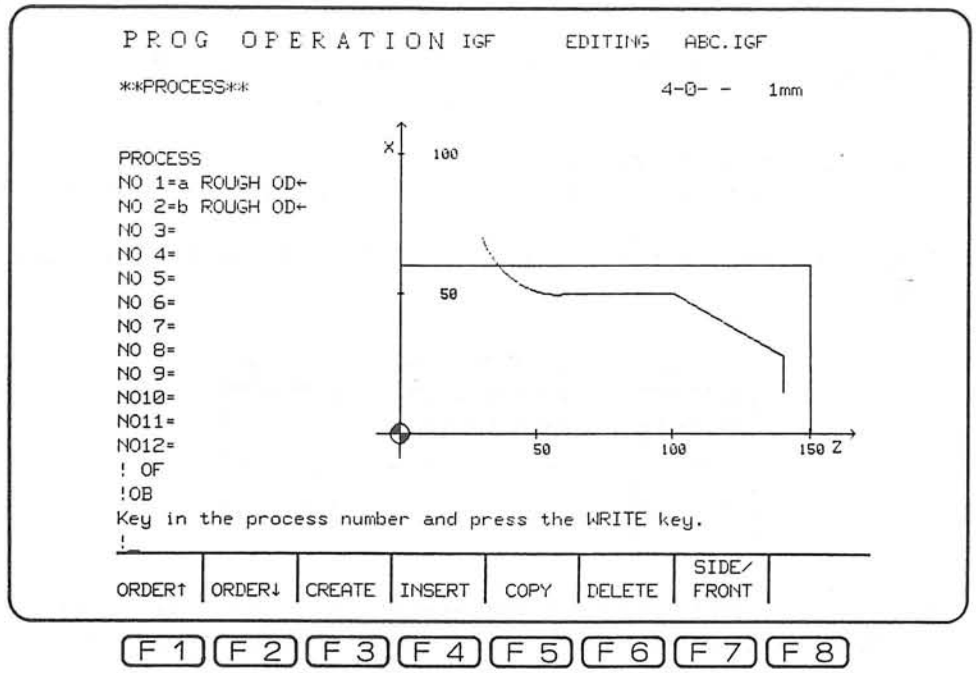
Rough cutting
Finish cutting
Copying
Groove cutting



Designate cutting directions first and then designate tool data for A and B turrets. Note that cutting conditions are common to both A and B.

3) Confirm the balance cut process.

a and b show that it is a balance cut process.



Precautions:

(1) When the control is unable to store the process of the other side because no space is found in the process chart when a balance cut process is designated during process designation, it displayed the following message at the console line and the process is made as a single (A-turret side) process.

Memory area for B-turret process is getting full.

(2) In case the single process already made needs to be changed as a balance cut process during process designation:

When the process on the A-turret side is to be changed,

Turret AB= Ø ⟶ 2

The process is changed as the balance cut process of the A-turret side. However, tool data for the B-turret side needs to be input.

When the process on the B-turret side is to be changed,

Turret AB= 1 ⟶ 2

The process is changed as the balance cut process of the B-turret side. However, tool data for the A-turret side needs to be input.

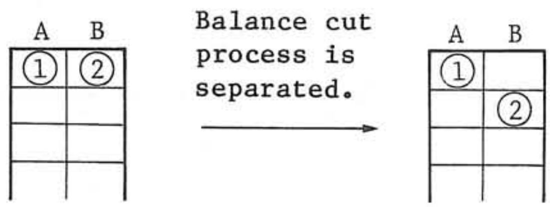
(3) When the turret designation is changed to "Ø" or "1" during process designation for the balance cut process already made, the process is changed as the single process for A- or B-turret side.

Turret AB= 2 $\rightarrow$ Ø Single process for the A-turret side
$\searrow$ 1 Single process for the B-turret side

(4) When cutting conditions or shape data are changed, the process of the other turret side is also modified.

(5) When balance cut process is deleted, the process of the other side turret remains as a single process of either A- or B-turret.

(6) When the balance cut process is to be separated as a result of process test:



Processes 1 and 2 become independent processes for A and B turrets automatically. If processes 1 and 2 are put together, they become a balance cut process again.

(7) When conditions are modified:

Conditions on the other turret side are also modified as in the process making.

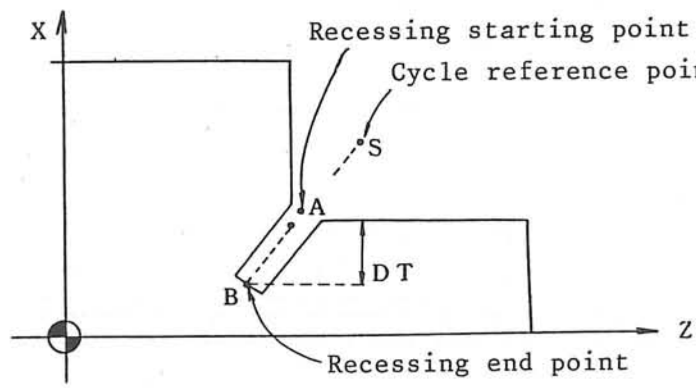
4-6. RECESSING

Parameters used for defining recessing process are indicated below:

REF. PT	X=	Reference point X (in diameter)
	Z=	Reference point Z
START PT	XA=	Recessing starting point X (in diameter)
	ZA=	Recessing starting point Z
END PT	XB=	Recessing end point X (in diameter)
	ZB=	Recessing end point Z
DEPTH	DT=	Recessing depth*

* Effective when XB, ZB are not specified.

The recessing definition requires at least the parameters XA, ZA, and either XB and ZB or DT.

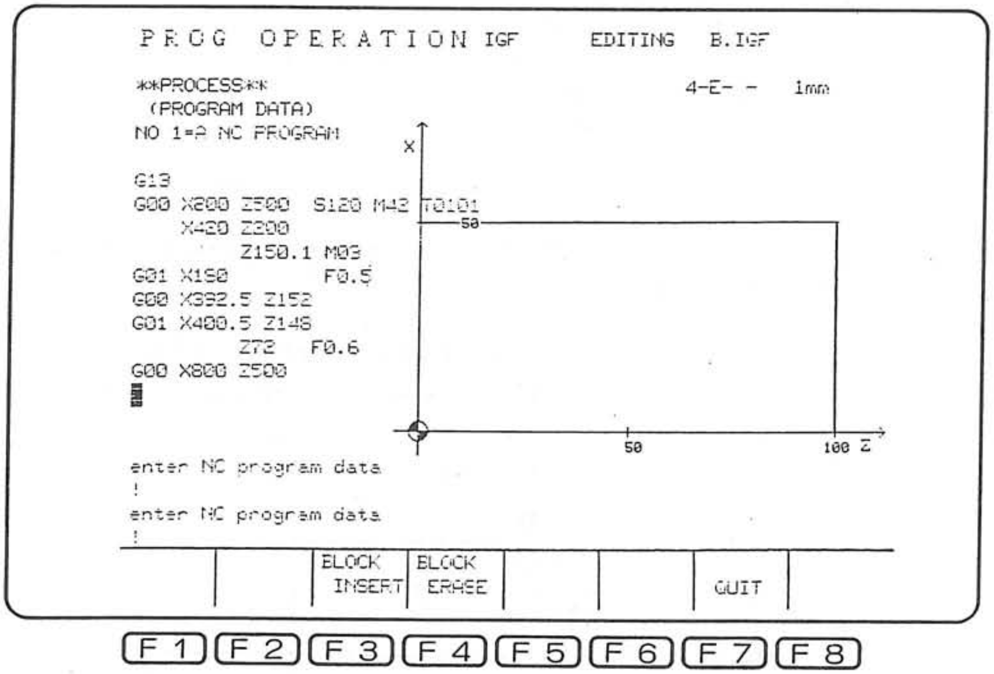


(1) When parameter DT is specified, recessing is made in 45° angle direction from point A up to the depth of $\sqrt{2}$ x DT.

(2) Parameters XB and ZB allow definition of recessing in a required direction. Recessing starts from point S.

4-7. NC PROGRAM INPUT

The designation of PROCESS NO 8 (NC PROGRAM) allows the insertion of blocks containing any desired G codes, M codes, CALL statements, etc. between the processes.



(1) The program to be input will initially be displayed on the console line area at the bottom of the display screen. On the completion of one input block, press the WRITE key to transfer to the display of the input just made to the console area at the upper part of the screen. This concludes input.

(2) To input more than one block (as on the display screen shown above), use the cursor key on the console panel to move the cursor down and input the next data block.

(3) If the entire block will not fit on the screen at one time, use the PAGE key on the operational panel to advance to the next display screen, then continue the input process. (The screen will not advance automatically.)

(4) The maximum number of characters permitted for one program during NC program input is 1Ø24.

Note: Both the space (blank) bar and the carriage return (WRITE key) are counted as characters. This should be kept in mind when determining the length of programs.

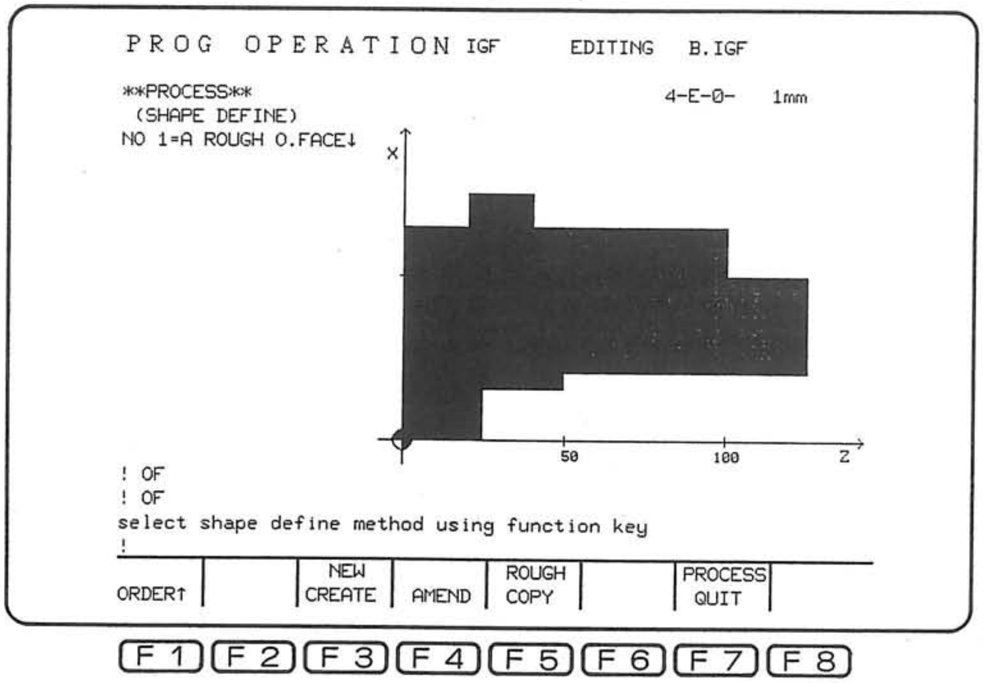
(5) Before conducting PROCESS NO 8 (NC PROGRAM), be certain that the coolant is off and that the turret has been retracted to the rotation position.

(6) Pressing function key [F3] (BLOCK INSERT) will open one empty line at the cursor position.

(7) Pressing function key [F4] (BLOCK ERASE) will remove the line at which the cursor is currently located.

SECTION 5 EDITING DEFINED SHAPES

In the SHAPE DEFINE process, editing of the shape elements and copy of shapes in units of processes are possible. Function keys [F4] (AMEND) and [F5] (ROUGH COPY) are used for this editing operation.



5-1. AMEND

Editing of the shape elements while observing the displayed shape is possible.

After the editing, shapes before and after the editing operation can be compared through the graphic display. With the color graphic specification, the shape after the editing is displayed in magenta color.

5-2. ROUGH COPY

Shapes defined in the previous processes are copied in the "finish" process. If the ROUGH COPY function is specified from a process other than finish, an error occurs (rough copy command).

SECTION 6 DEFINING ARBITRARY SHAPE

6-1. START POINT ENTRY

The CRT first displays the prompts for the entry of the start point coordinates. Note that these prompts are displayed only for the first shape element.

START PT	SX=	Start point X (in diameter)
	SZ=	Start point Z

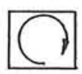
Enter the coordinates of the start point.

6-2. SHAPE ELEMENT AND SHAPE ELEMENT DATA ENTRY

(1) Prompts Displayed for Shape Element Entry

a) Lines ($\boxed{\leftarrow}$ $\boxed{\rightarrow}$ $\boxed{\uparrow}$ $\boxed{\downarrow}$ $\boxed{\nwarrow}$ $\boxed{\swarrow}$ $\boxed{\nearrow}$ $\boxed{\searrow}$)

SUR. RGH	SR=	Number of "▽" marks representing the surface roughness
END PT	X =	End point X (in diameter)
	Z =	End point Z
PRE TAN	TL=	Tangent to previous shape
NEXT TAN	TN=	Tangent to next shape
Z ANGLE	A =	Angle in reference to Z-axis (deg.)

b) Circles ( )

SUR. RGH	SR=	Number of "▽" marks representing the surface roughness
END PT	X =	End point X (in diameter)
	Z =	End point Z
PRE TAN	TL=	Tangent to previous shape
NEXT TAN	TN=	Tangent to next shape
RADIUS	R =	Radius
CENTER	CX=	Arc center X (in diameter)
	CZ=	Arc center Z

c) Chamfering ($\boxed{C}$)

SUR. RGH	SR=	Number of "▽" marks representing the surface roughness
CHF SIZE	C =	Chamfer size

d) Chamfering ($\boxed{R}$)

SUR. RGH	SR=	Number of "▽" marks representing the surface roughness
CHF SIZE	R =	Chamfer radius

A total of 48 shape elements can be defined.

(2) Entry of Data in Response to Prompts and Determination of Shape Elements

a) All prompts indicated above are not always displayed for lines and circles (arcs). In addition, entry of data is necessary only for the prompts which will define the shape element in question.

b) The IGF system refers the data from the top line for defining the shape element. At the point the shape can be defined from the data having been referenced, the IGF system stops further processing for the shape element being defined. This means that the data excessively entered is ignored by this system.

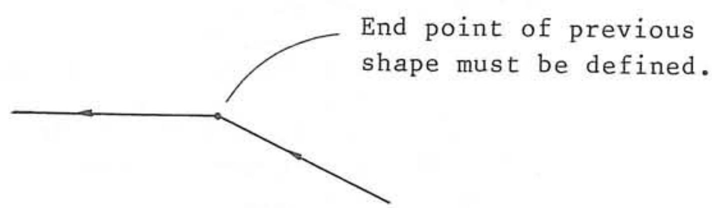
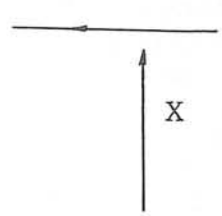
Note: Although the prompt "SUR RGH SR=" is displayed for all shape elements, it is not necessary for the definition of a shape elements.

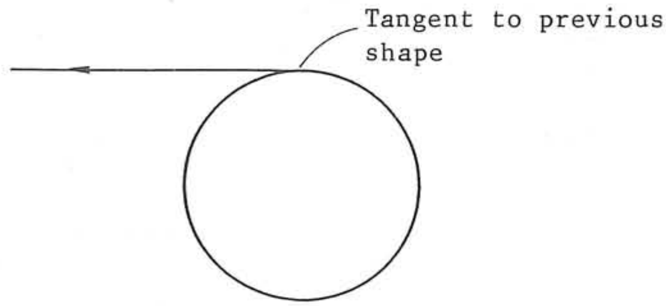
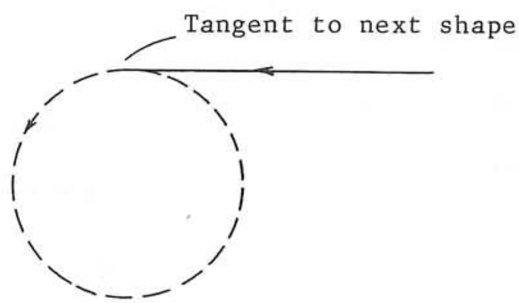
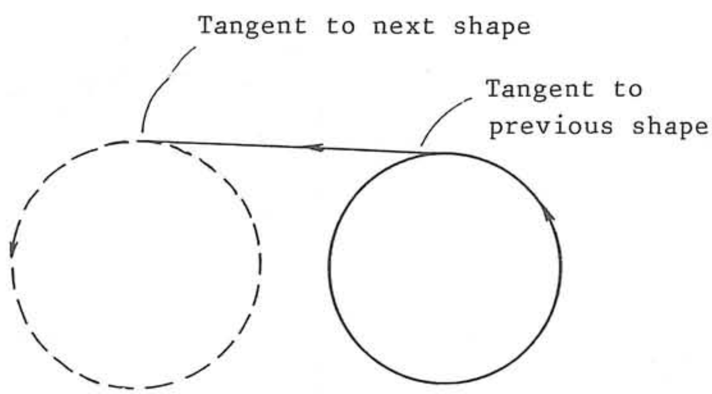
In addition, even if a shape element is defined, the defined shape element is not always displayed on the CRT.

c) Combination of data sets necessary for defining shape elements:

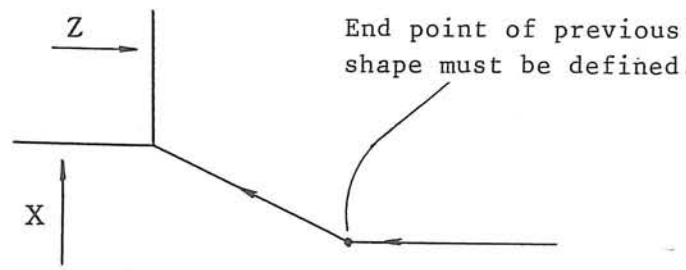
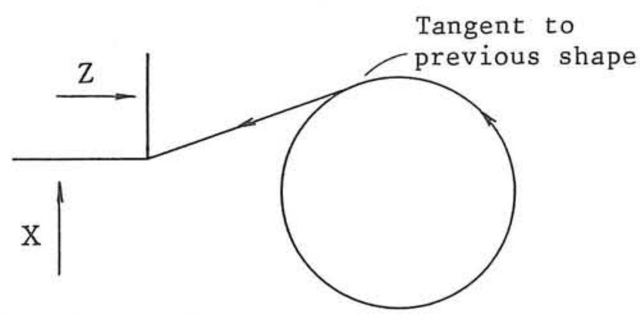
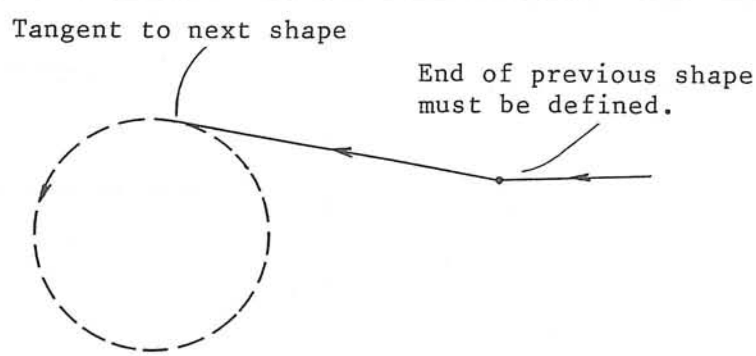
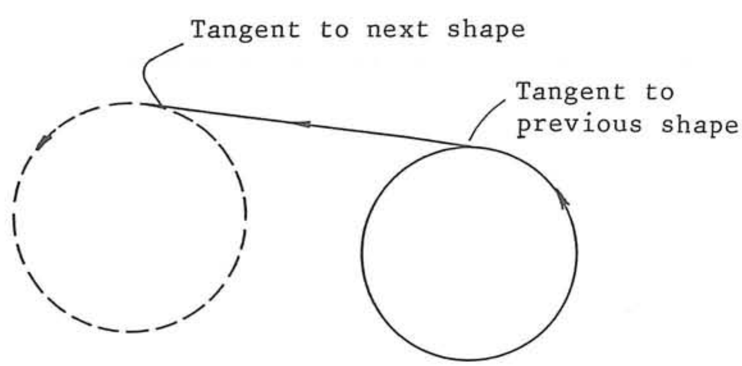
The charts below indicate the combination of the data sets required for defining shape elements.

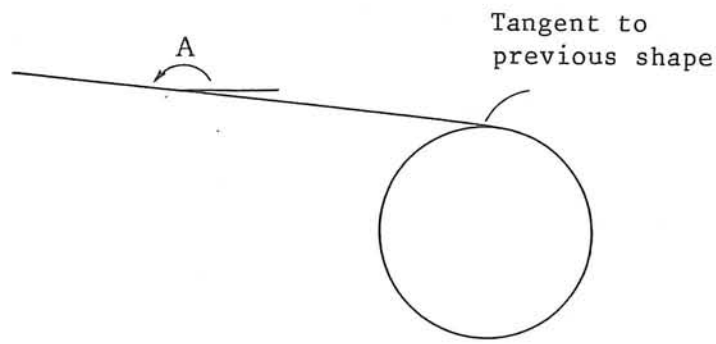
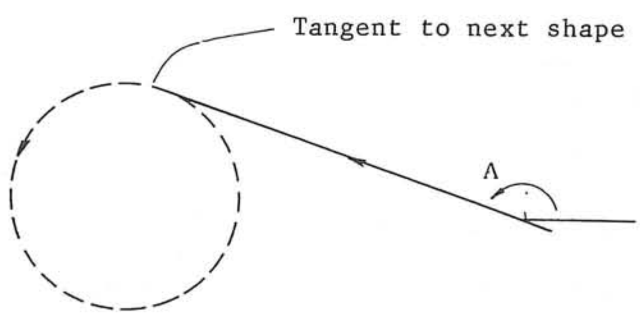
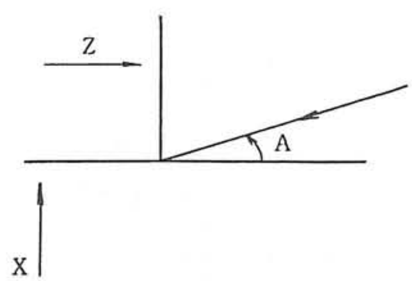
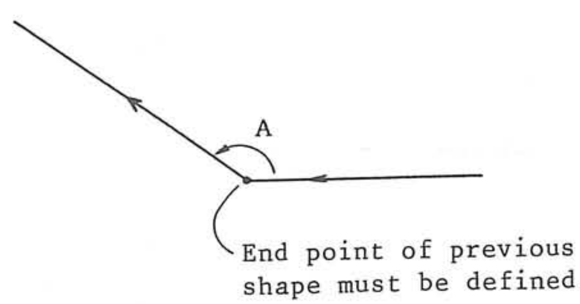
Lines Parallel to Axis: ($\boxed{\leftarrow}$ $\boxed{\rightarrow}$ $\boxed{\uparrow}$ $\boxed{\downarrow}$)

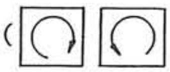
Combination of Input Data	Supplement
(1) No entry	
(2) END PT X (Z)	

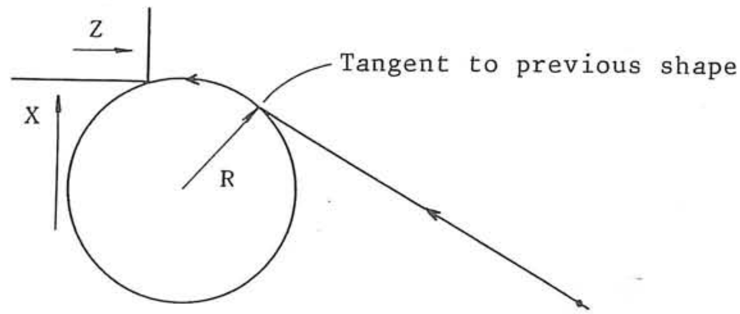
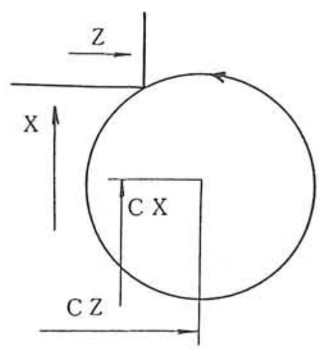
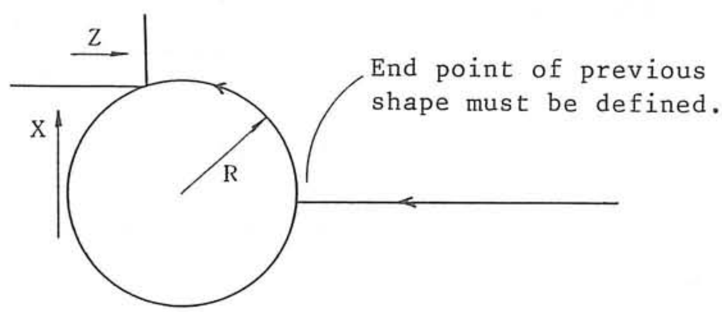
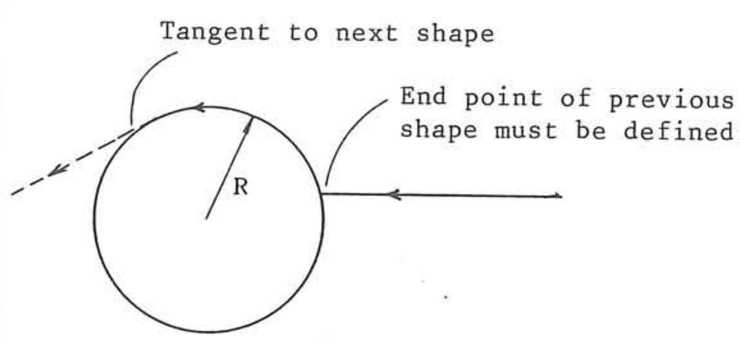
Combination of Input Data	Supplement
(3) PRE TAN TL	
(4) NEXT TAN TN	
(5) PRE TAN TL NEXT TAN TN	

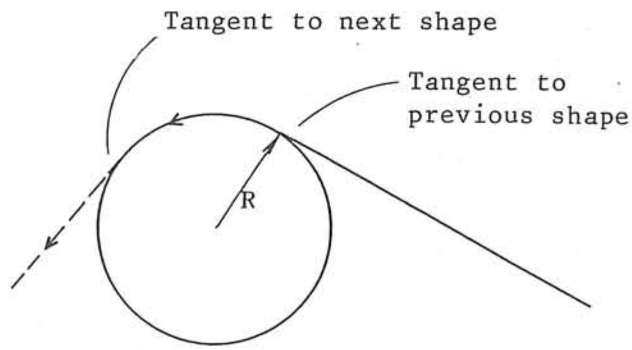
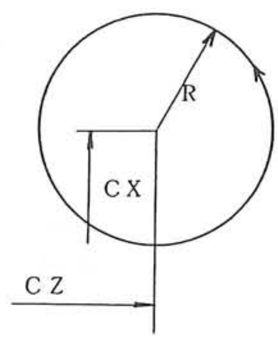
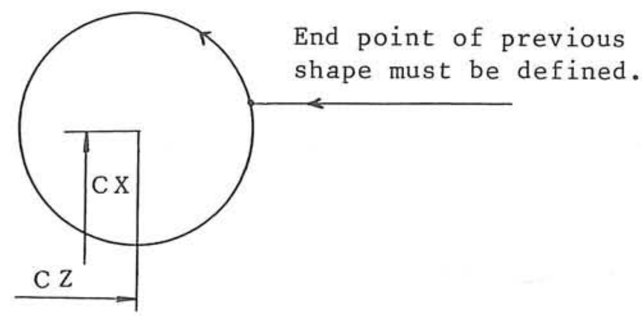
Lines not Parallel to Axis (Taper): (↖ ↙ ↗ ↘)

Combination of Input Data	Supplement
(1) END PT X END PT Z	
(2) END PT X END PT Z PRE TAN TL	
(3) NEXT TAN TN	
(4) PRE TAN TL NEXT TAN TN	

Combination of Input Data	Supplement
(5) PRE TAN TL Z ANGLE A	
(6) NEXT TAN TN Z ANGLE A	
(7) END PT X END PT Z Z ANGLE A	
(8) Z ANGLE A	

Arcs: ()

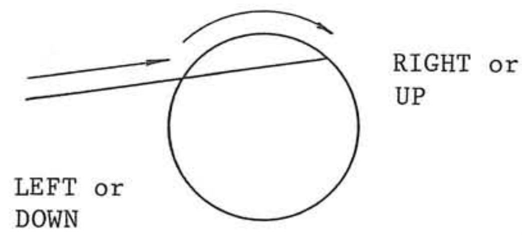
Combination of Input Data	Supplement
(1) END PT X END PT Z PRE TAN TL RADIUS R	
(2) END PT X END PT Z CENTER CX CENTER CZ	
(3) END PT X END PT Z RADIUS R	
(4) NEXT TAN TN RADIUS R	

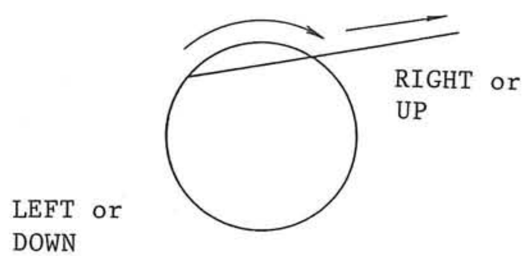
Combination of Input Data	Supplement
(5) PRE TAN TL NEXT TAN TN RADIUS R	
(6) RADIUS R CENTER CX CENTER CZ	
(7) CENTER CX CENTER CZ	

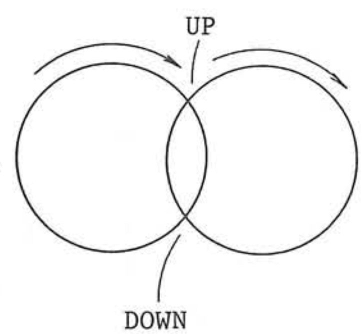
6-3. SELECTION OF START POINT (→=1, ←=2, ↑=3, ↓=4)

These prompts request the operator to select the cross point to be defined when shape elements are defined in the combination of line and arc, arc and line, and arc and arc. In these cases, two cross points will be present unless they are not tangent each other.

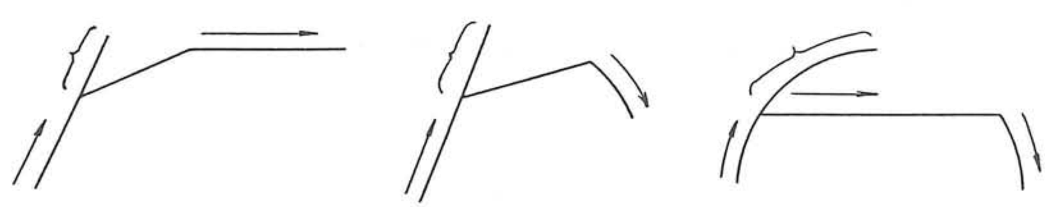
→	= 1	RIGHT
←	= 2	LEFT
↑	= 3	UP
↓	= 4	DOWN







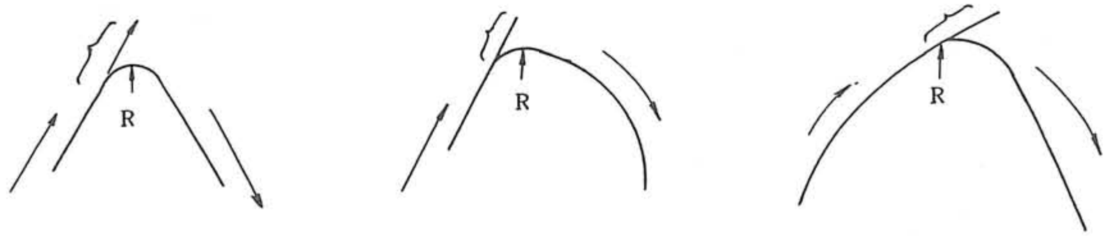
6-4. C-CHF DEFINITION



Sections indicated by the symbol "{" are not displayed unless the end point is not given.

If two cross points are present between the shape elements preceding and following the C-CHF command, the prompt for selecting the start point is given. The chamfering is defined at the selected cross point.

6-5. R-CHF DEFINITION



Sections indicated by the symbol "{ " are not displayed unless the end point is not given.

Use of and [R-CHF] for defining arcs:

There are two types of shape elements for defining arcs as indicated above. These two types of shape elements should be used depending on the process of shape definition.

a) [R-CHF] is used for defining an arc having known radius under the condition that it is tangent to

   i) preceding and following lines
   ii) preceding and following arc and line
   iii) preceding and following arcs

b) Note that if the preceding and following shape elements have two cross points, the arc is defined at the cross point selected in response to the prompt given.

c) Use or when the coordinates of the center of the arc and those of the start and end point of the arc.

6-6. SHAPE DEFINITION CAN'T BE DONE MESSAGE

Message "shape definition can't be done" might be displayed during the shape definition process. This indicates that the entered data is insufficient for defining the shape element or inconsistent shape definition data has been entered. In this case, the cursor is returned to a shape element from which the data should be entered again.

6-7. TO DELETE SHAPE ELEMENT DATA AFTER IT HAS BEEN INPUT

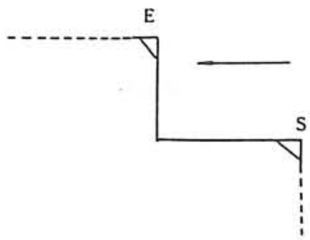
The key operation below deletes the data.

! N [WRITE]

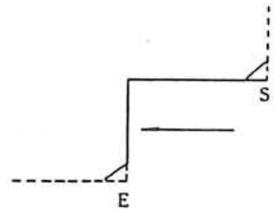
6-8. PROCESSING OF START AND END POINTS FOR C-CHF

When the C-CHF is defined as the first or the final shape, processing of this shape element depends on the direction of cut selected.

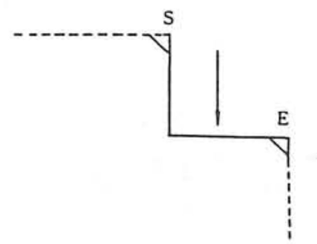
1. OD ←



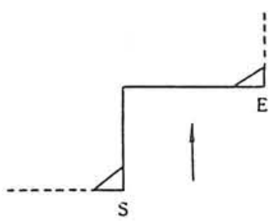
2. ID ←



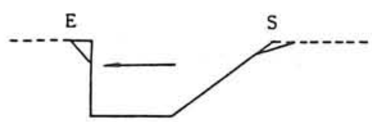
3. O.FACE ↓



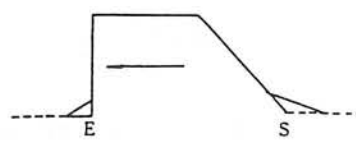
6. I.FACE ↑



7. M.OD ←

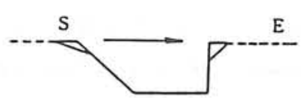


8. M.ID ←

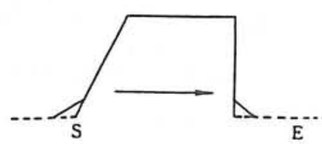


9. M.FACE ↓

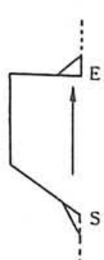
1Ø. M.OD →



11. M.ID →

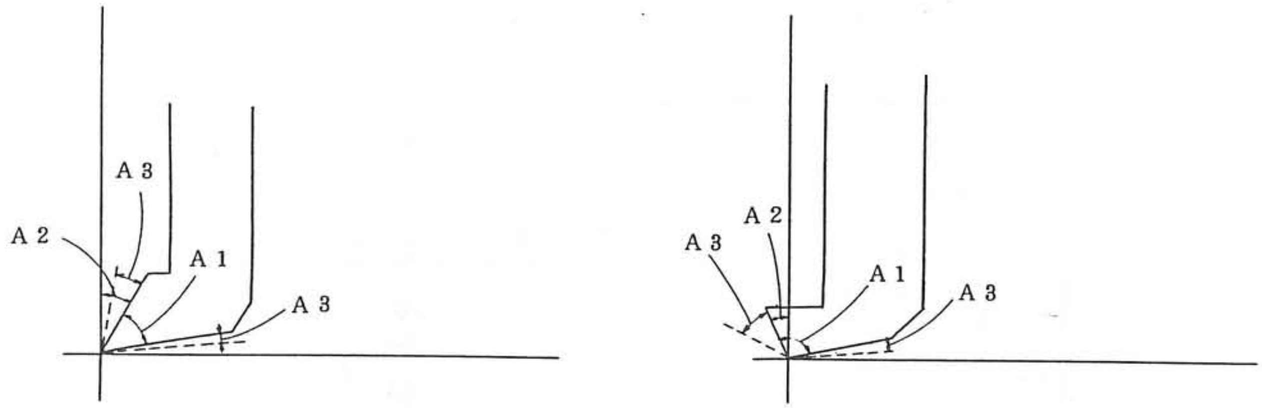


12. M.FACE ↑



6-9. TOOL EDGE SHAPE CHECK

(1) Tool Edge Shape Check



TOOL ANGLE A1 ($Ø \leq$ A1 $\leq$ 36Ø°)
EDGE ANGLE A2 (-36Ø° $\leq$ A2 $\leq$ 36Ø°)
MARGIN ANGLE A3 ($Ø \leq$ A3 $\leq$36Ø°)

⟵ : Direction of cuts

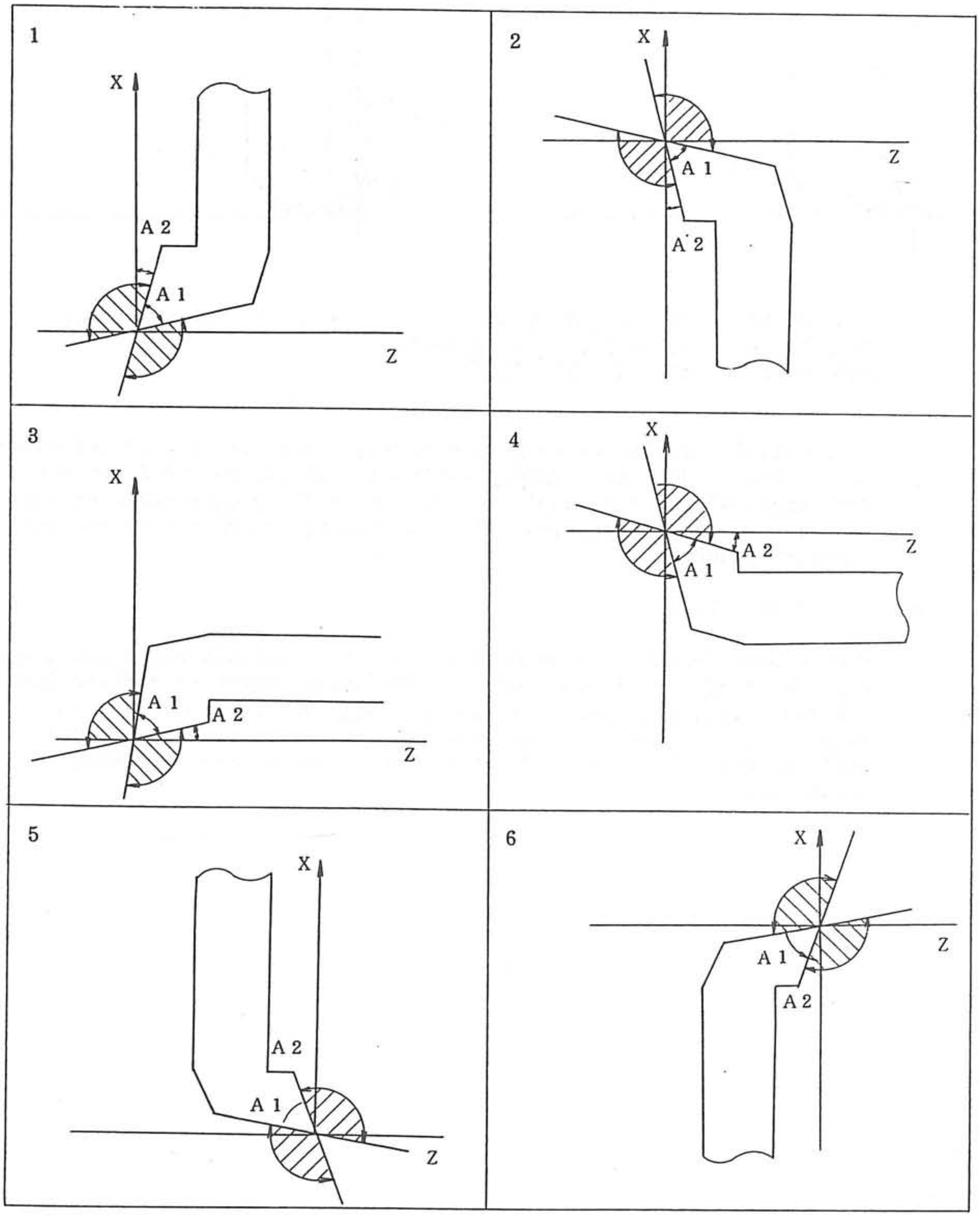
When defining arbitrary shapes, tool edge check is carried out based on the (TOOL ANGLE A1 + MARGIN ANGLE A3 x 2) and the EDGE ANGLE A2. Tool angle A1 and edge angle A2 are set in the graphic-edit process or tool data setting operation. The margin angles are set by the IGF integer parameter.

(2) Tool Shape Check

When shape element is commanded in the shape definition, interference with the tool edge is checked. If the shape cannot be defined due to interferences, the error message is displayed and the cursor is located at the shape element, which has caused an error. Whether the tool shape check is carried out or not is set by the IGF integer parameter.

Tool shape check patterns are as indicated below. The range acceptable for shape definition is indicated by hatching lines.

A1: TOOL ANGLE
A2: EDGE ANGLE



6-1Ø. ADJOINT SHAPE INPUT FUNCTION

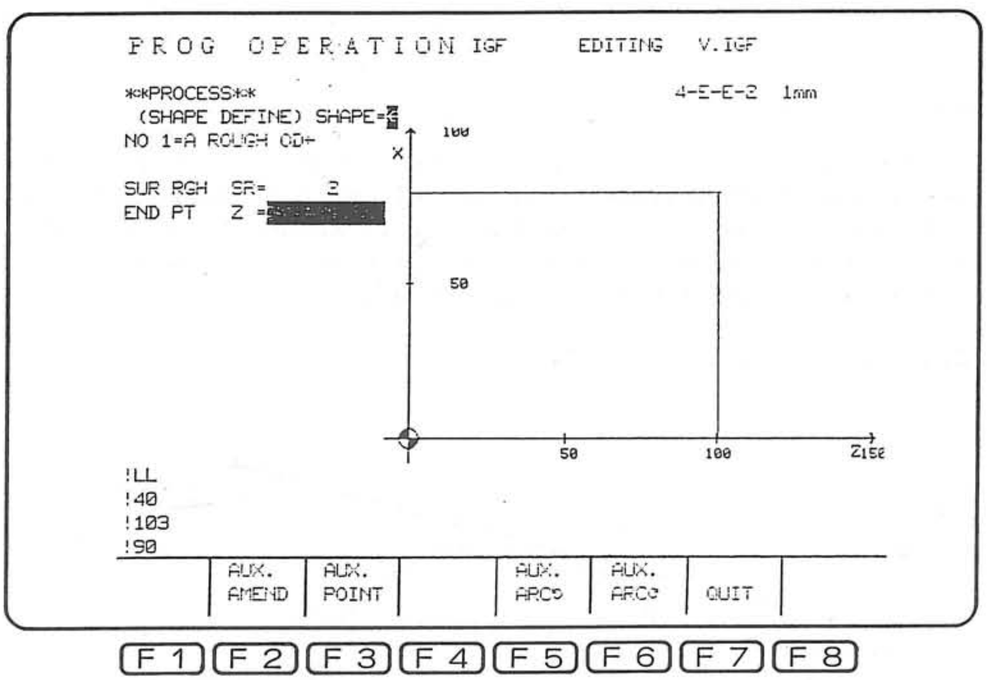
The adjoint shape input function is newly added to the system. This function allows the input of an adjoint shape when the shape to be defined is complicated and calculation of the end point coordinates or cross-point coordinates is very difficult to make the impossible calculation possible, thus allowing the definition of required complicated shape.

(1) Types of Adjoint Shapes

Adjoint shape
- Adjoint point
- Adjoint line
- Adjoint arc
  - (CCW)
  - (CW)

(2) Adjoint Shape Input Screens

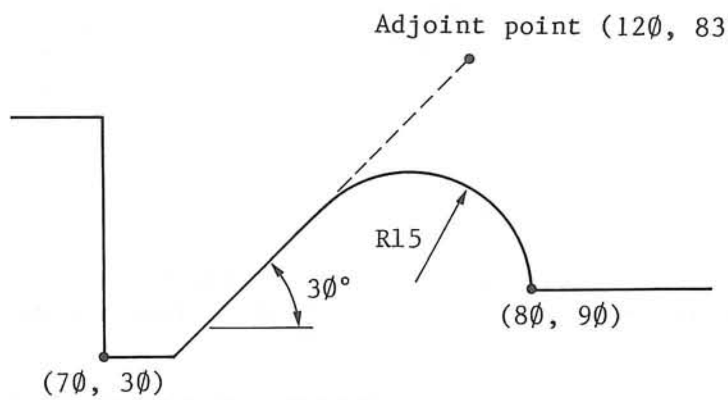
Input of an adjoint shape is possible from the shape definition screen (SHAPE DEFINE page) using a corresponding function key.



(3) Adjoint Point

See Example 1. In this shape definition, the end point of a taper cannot be calculated since the coordinates of the arc center are not known and therefore shape definition is impossible. However, if the coordinates of a point lying on an extension the taper is input, calculation of the coordinates of the arc center becomes possible. This point is called the "adjoint point".

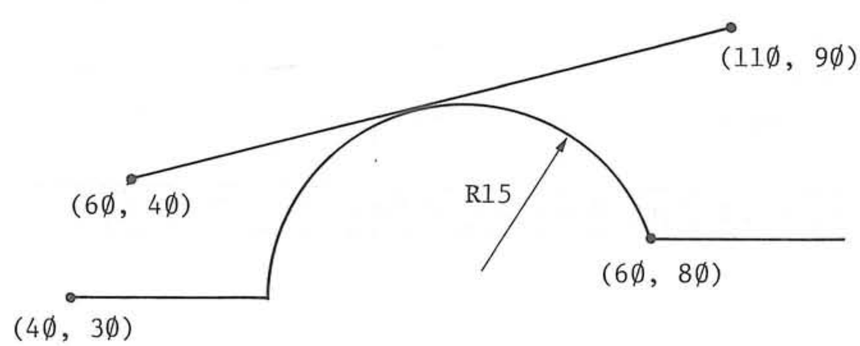
Example 1:



(4) Adjoint Line

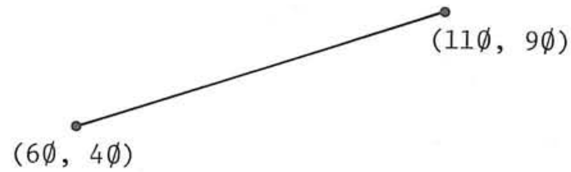
See Example 2. In this case, as in Example 1 in (3), definition of the shape is impossible since the center coordinates of the arc are not known. However, if the tangent is known, then the calculation of the center coordinates of the arc is possible. This line (tangent to the arc) is called the "adjoint line".

Example 2:

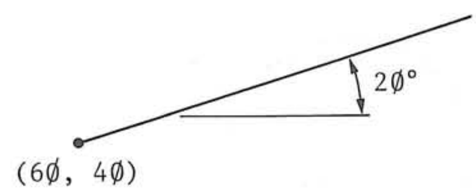


* Adjoint lines are input in the following two methods:

a) Input of the coordinates of the two points on the line



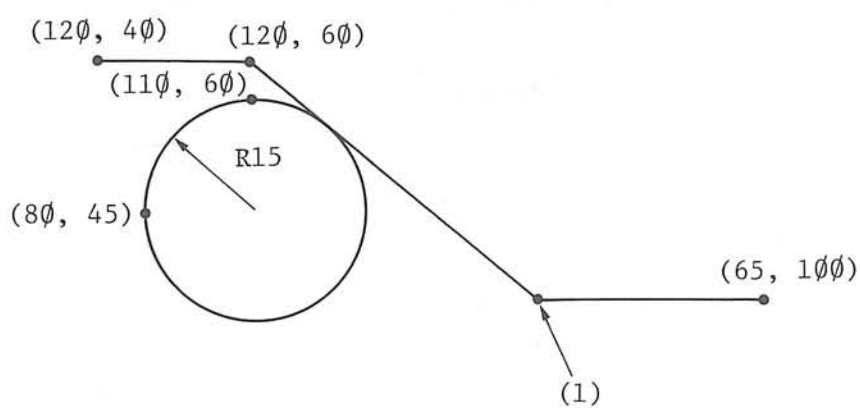
b) Input of the coordinates of a point on the line and an angle to Z-axis



(5) Adjoint Arc

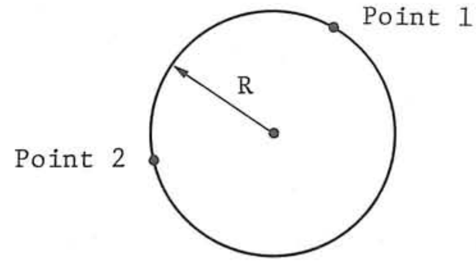
See Example 3. Usually, definition of a shape is impossible if the coordinates of point (1) in the figure are not known. However, when an arc contacting to the taper in question is known, it is possible to calculate the coordinates of point (1). This arc is called the "adjoint arc".

Example 3:

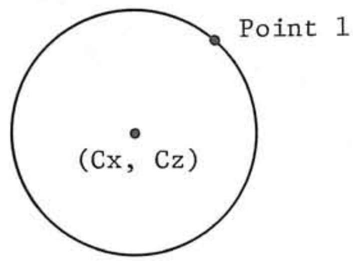


* Adjoint arcs are input in the following three methods:

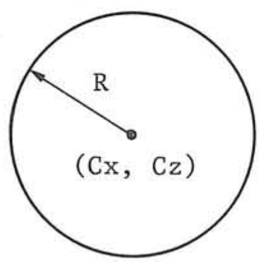
a) Arc having radius R and passing points 1 and 2 (as in Example 3)



b) Arc having the center at (Cx, Cz) and passing point 1

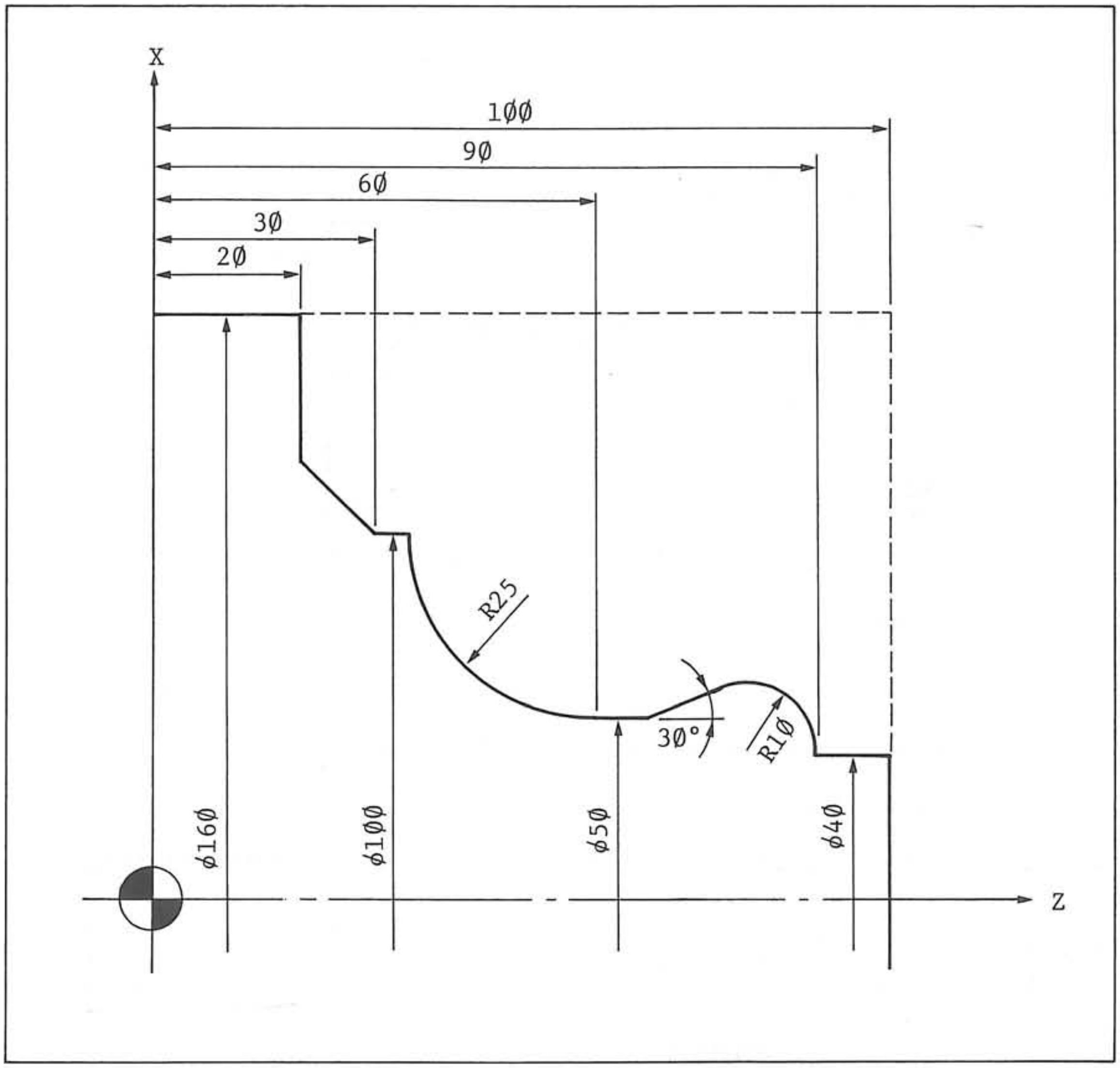


c) Arc having the center at (Cx, Cz) with radius R

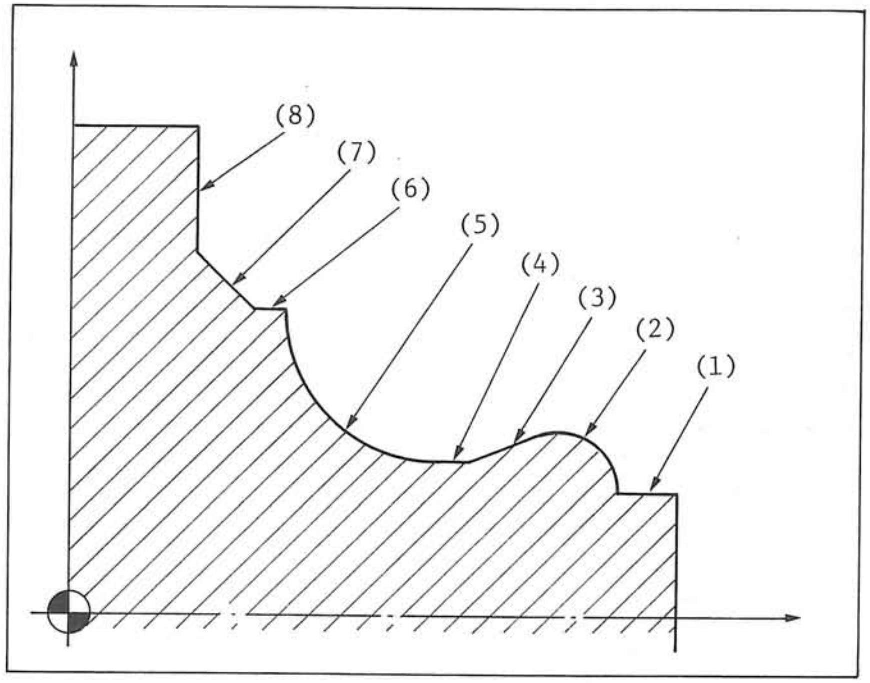


(6) Use of Adjoint Shapes

How the adjoint shapes are used in actual programming is explained below using an example.



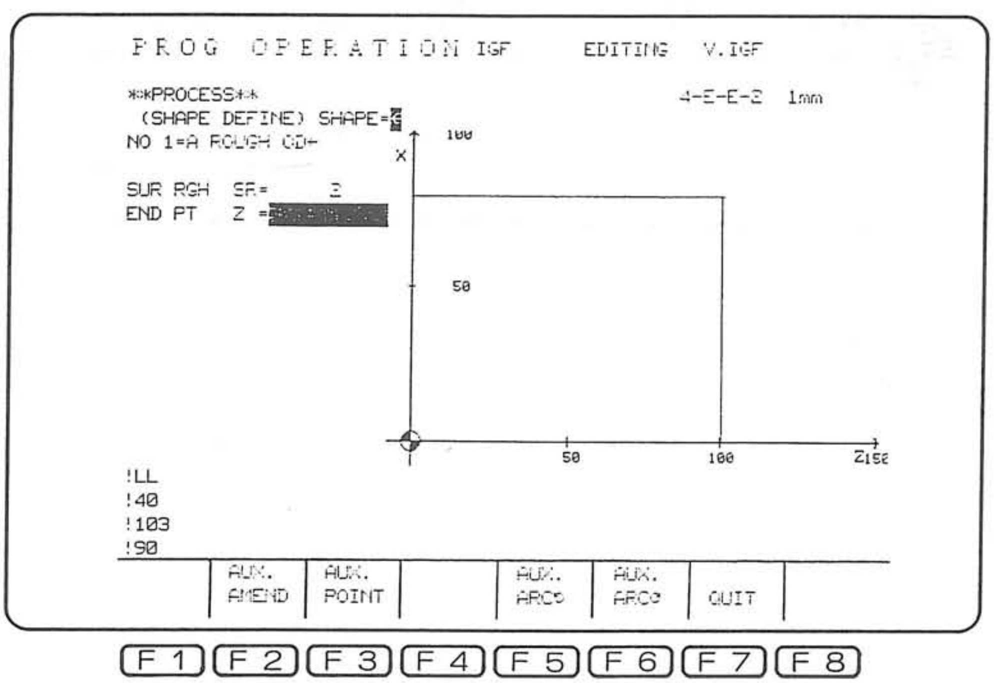
Since adjoint shapes are input from the SHAPE DEFINE page, explanation is provided below from the shape definition process.



1) Process (1)

Shape definition for process (1) does not use adjoint shapes since the shape definition is made for a longitudinal line whose coordinates of the start and end points are known.

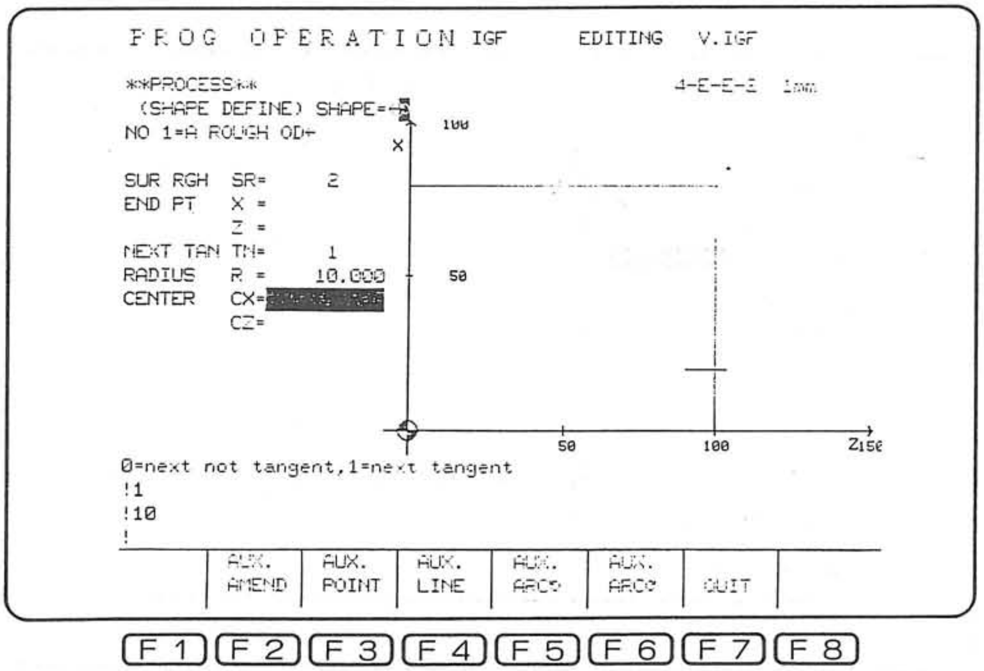
Input the data as indicated below.



2) Process (2)

Shape definition for process (2) is made by designating the arc which contacts to the next shape. This does not require adjoint shapes.

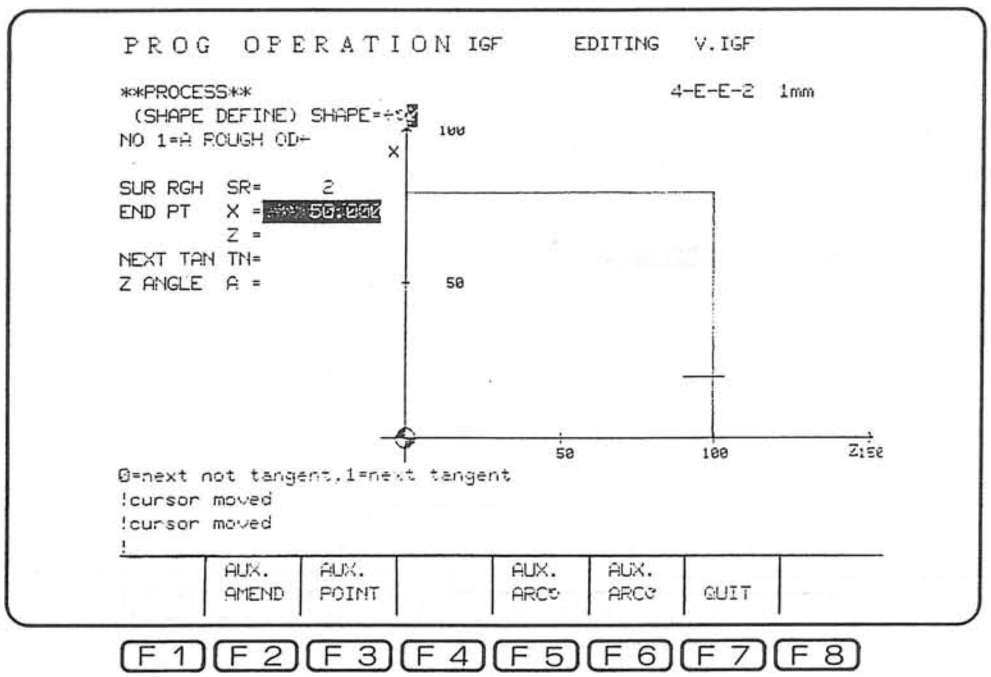
Input the data as indicated below.

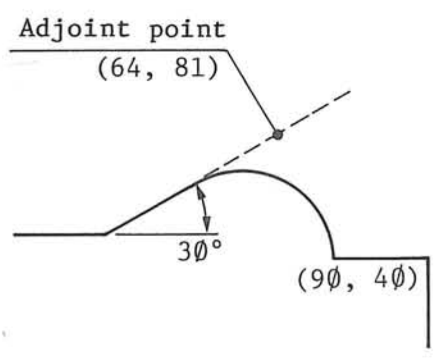


3) Process (3)

a) For process (3), taper should be defined. However, the coordinates of the start point and the end point (Z-axis) are not known from the drawing. Input of the adjoint point will allow the definition of the shape for process (3).

Follow the steps below to input the adjoint point.



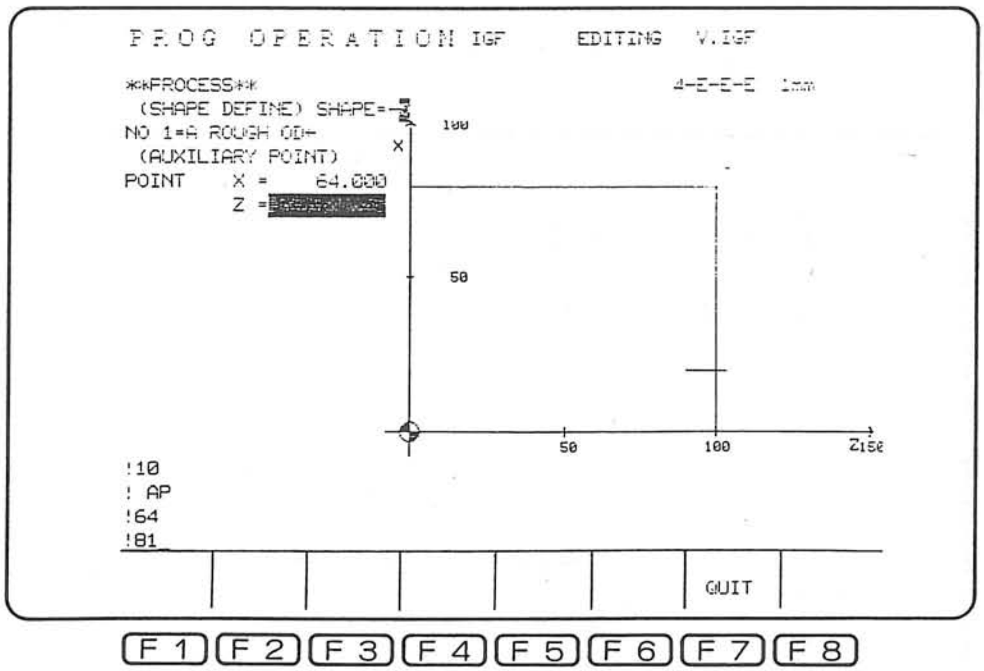


* Adjoint point may be taken at any point on the extension of the taper.

Press the function key [F3] (AUX. POINT). The screen will change as indicated below.

Input the coordinates of the adjoint point:

X = 64
Z = 81

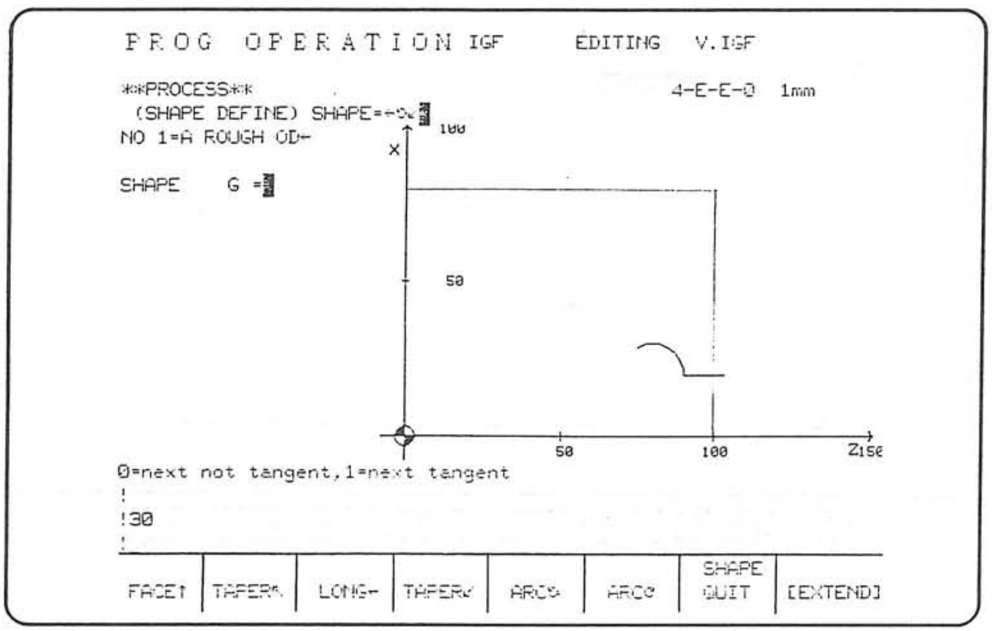


b) After the input of the adjoint point, input the data to define the taper. The data known from the drawing related with the taper to be defined is only the angle to Z-axis. Therefore, input the taper angle at this page.

A = 3Ø

Press the function key [F7] (QUIT).

After the input of the taper defining data, the arc for process (2) is drawn on the CRT screen.

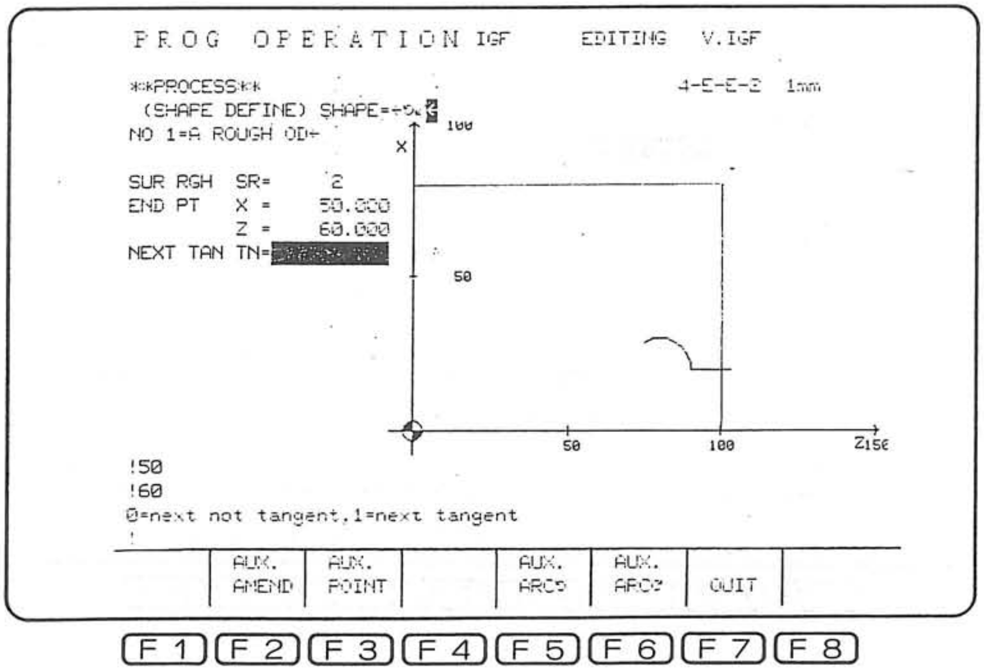


F 1 F 2 F 3 F 4 F 5 F 6 F 7 F 8

4) Process (4)

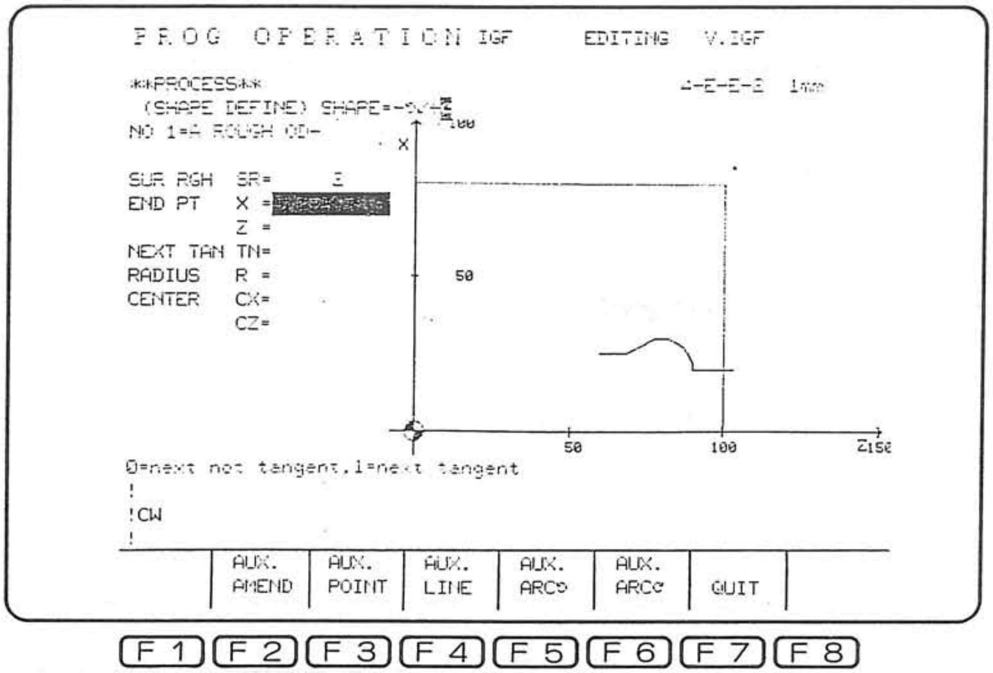
Shape definition for process (4) does not require adjoint shape since the shape definition is made for a longitudinal line whose coordinates of the end point are known.

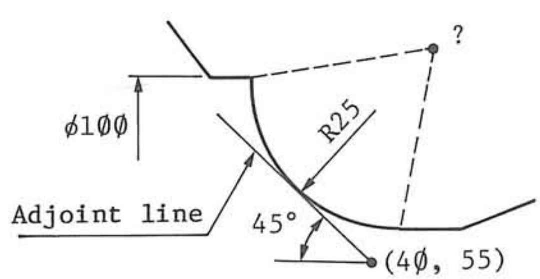
Input the data as indicated below.



5) Process (5)

a) The shape for process (5) is the clockwise arc, whose center coordinates are not known. Therefore, designate the adjoint line to define such a shape.



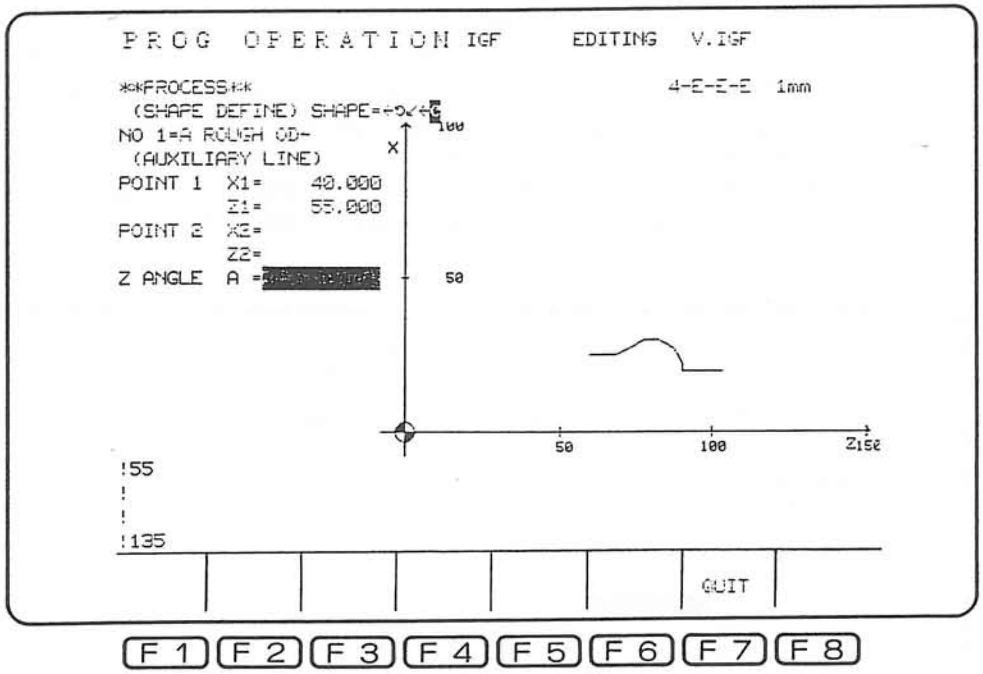


* An adjoint line can be defined by two points or one point and an angle.

Press the function key [F4] (AUX. LINE). The CRT screen will change as indicated below.

Input the data to define the adjoint line.

X1 = 4Ø
Z1 = 55
A = 135



Note: If coordinates of another point on the adjoint line (X2, Z2) are known, then input of the angle A is not necessary.

b) Input of the data to define the clockwise arc:

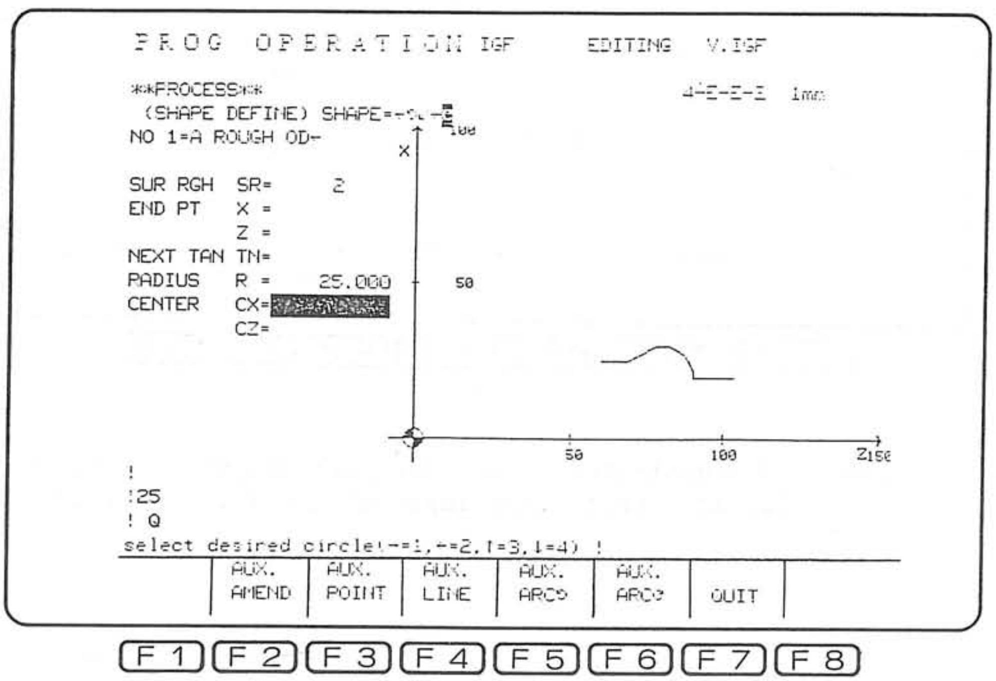
After the input of the adjoint line data, define the clockwise arc.

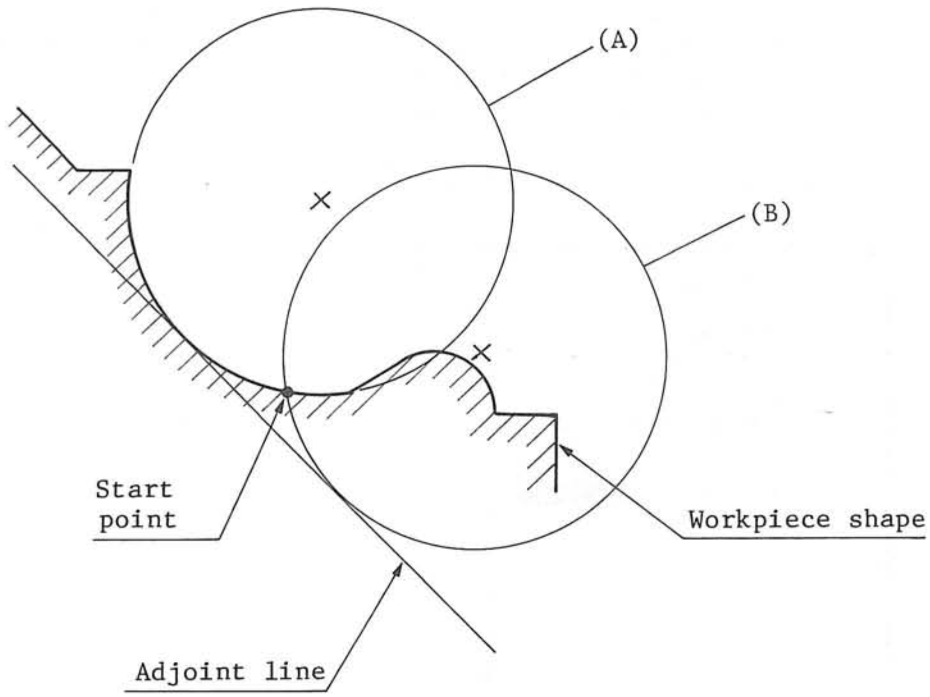
Since only radius R is known related with the arc, input it to define the arc.

R = 25

Press the function key [F7] (QUIT).

The CRT screen will display the prompt "select desired circle (→=1, ←=2, ↑=3, ↓=4)". This prompts allows the operator to select the arc to be defined if the input of the adjoint shape cannot define one arc. In this example, input of the adjoint line can define two possible arcs as shown below.



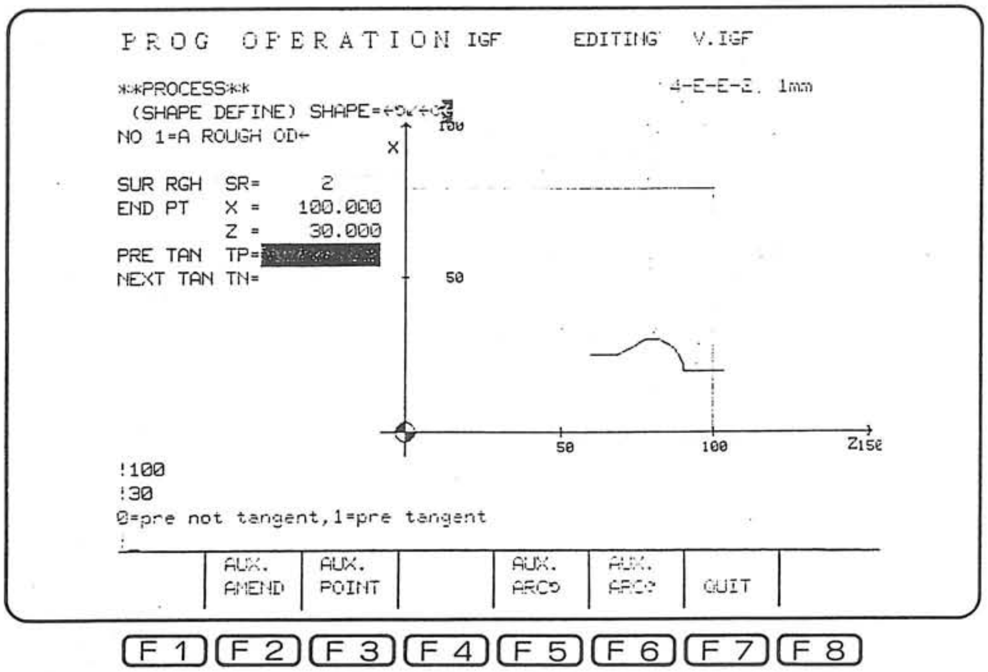


Since the arc to be defined is arc (A), selection of the arc should be made by inputting either "←=2" or "↑=3".

6) Process (6)

Shape definition for process (6) does not require adjoint shape since the shape definition is made for a longitudinal line whose coordinates of the end point are known.

Input the data as indicated below:

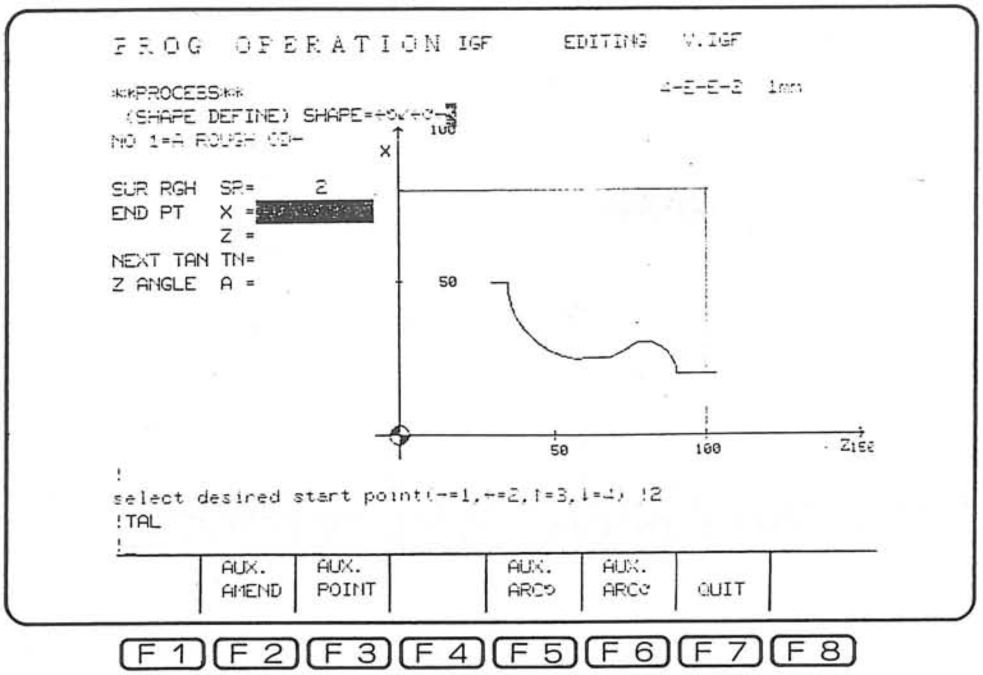


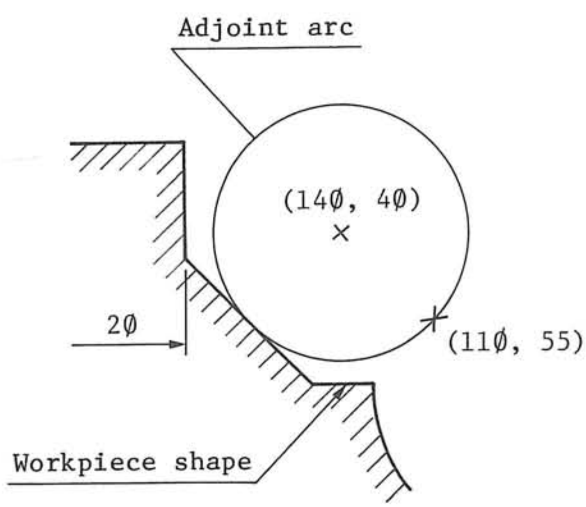
Note: After the input of the data for process (6), prompt "select desired start point (→=1, ← =2, ↑ =3, ↓ =4)" will be displayed. For the selection of the start point in response to this prompt, refer to 6-3, "Selection of Start Point".

7) Process (7)

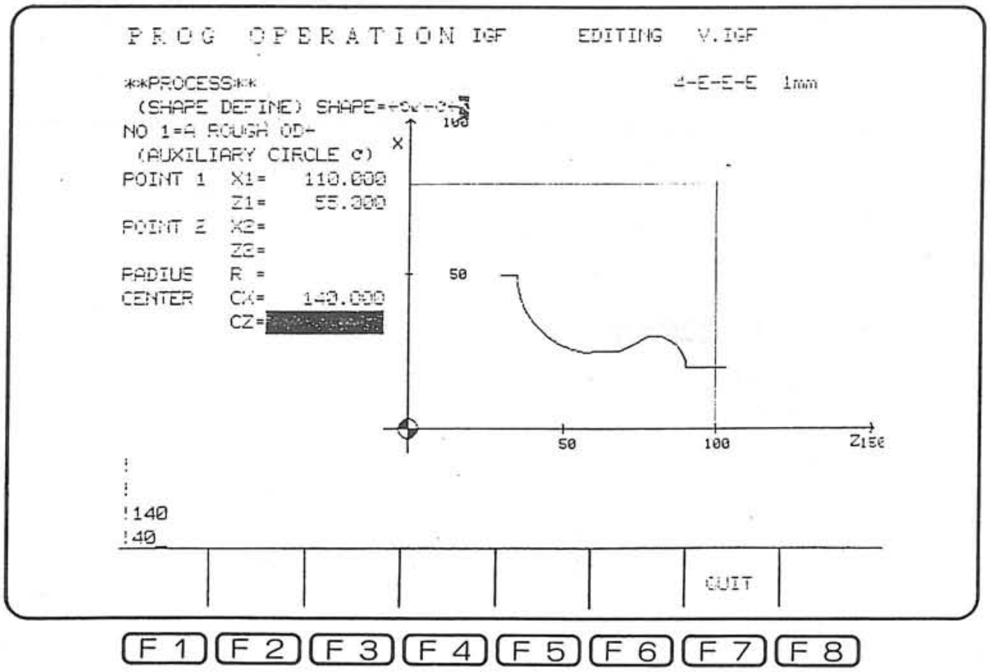
a) Input of adjoint shape (adjoint arc)

Process (7) requires the definition of a taper whose end point is not known. Therefore, designate the adjoint arc to define such a shape.





When the function key [F6] (AUX. ARC↻) is pressed, the display will change as indicated below, where the input of the data to define the adjoint arc is possible.

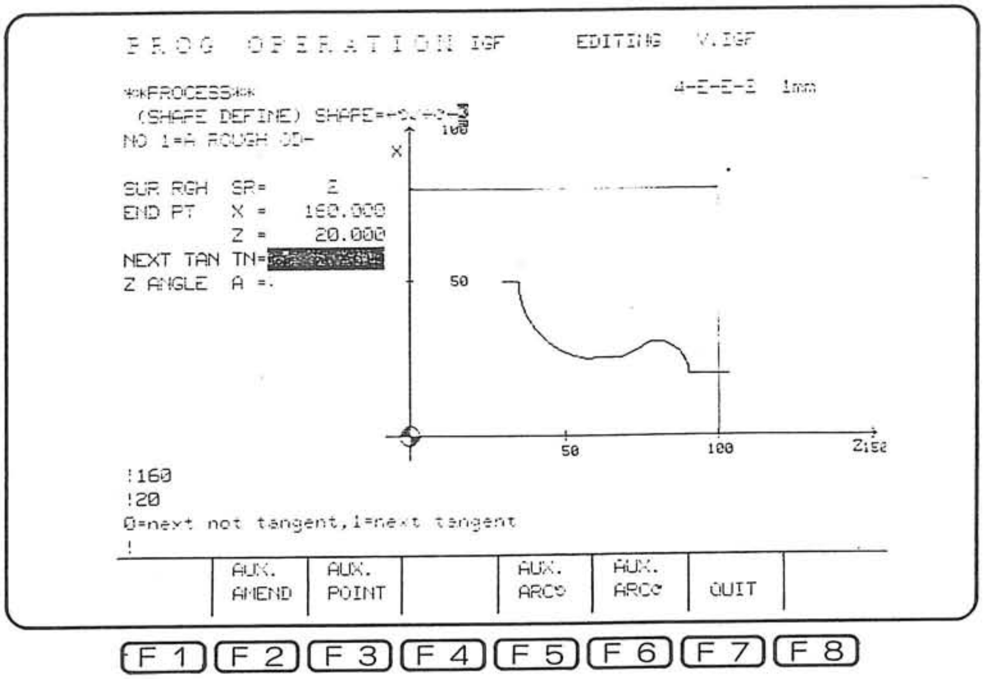


Note: Adjoint arc can be defined in any of three methods as explained in (5), "Adjoint Arc".

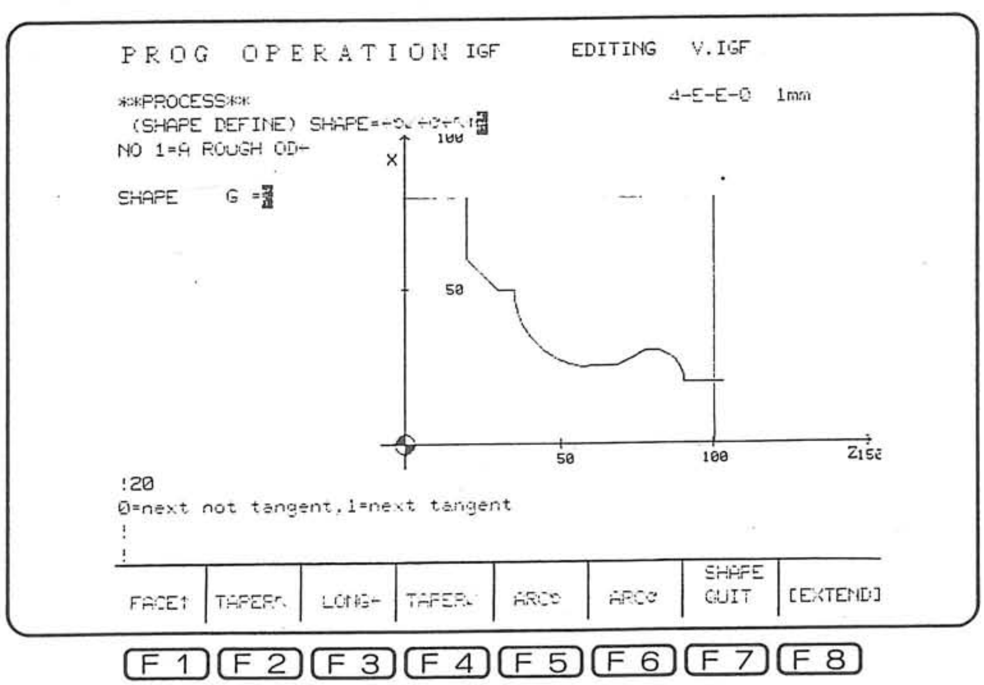
8) Process (8)

Shape definition for process (8) requires the definition of transverse line (line parallel to X-axis) whose end point is known. Therefore, input of the adjoint shape is not required.

Input the data as indicated below.



The steps above complete the input for shape definition.



(7) Precautions

If shape definition is possible only from the data input, data input to define the adjoint shape is ignored even if they are input. In such a case, the following message will appear on the CRT screen.

"auxiliary shape is not required."

SECTION 7 SHAPE REFERENCING FUNCTION (CROSS-POINT DISPLAY FUNCTION)

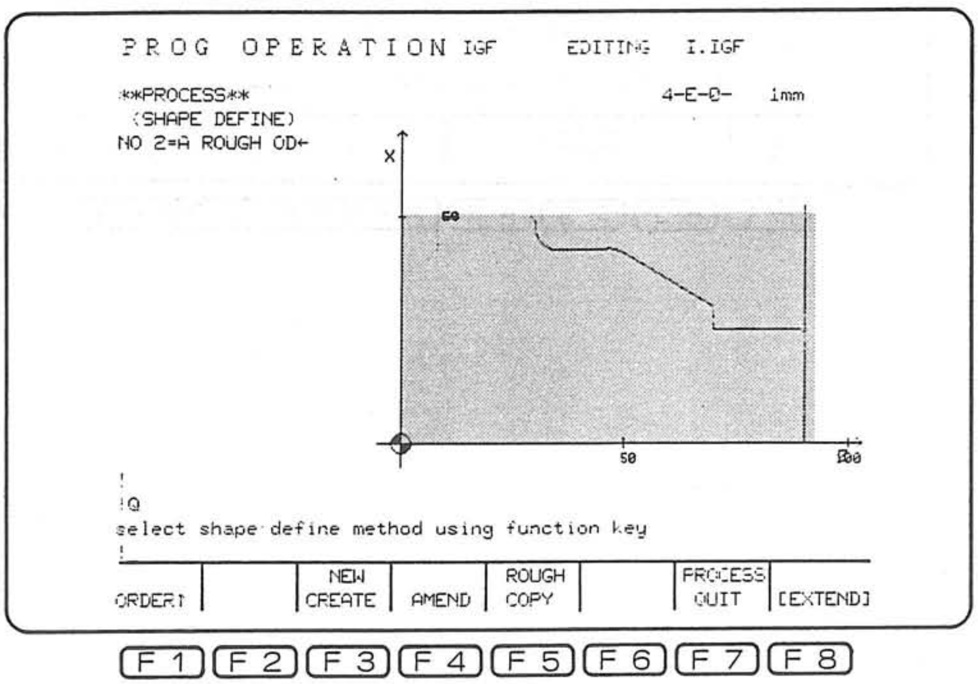
To define shapes, creation of a shape is possible by designating "tangent to previous shape" or "tangent to next shape", or inputting an adjoining shape even if the point of intersection is unknown.

The shape reference function calculates and displays the created point of intersection, radius, and center of an arc.

An example of how to use the shape reference function is explained below using the "ROUGH OD←" process explained in the BASIC section.

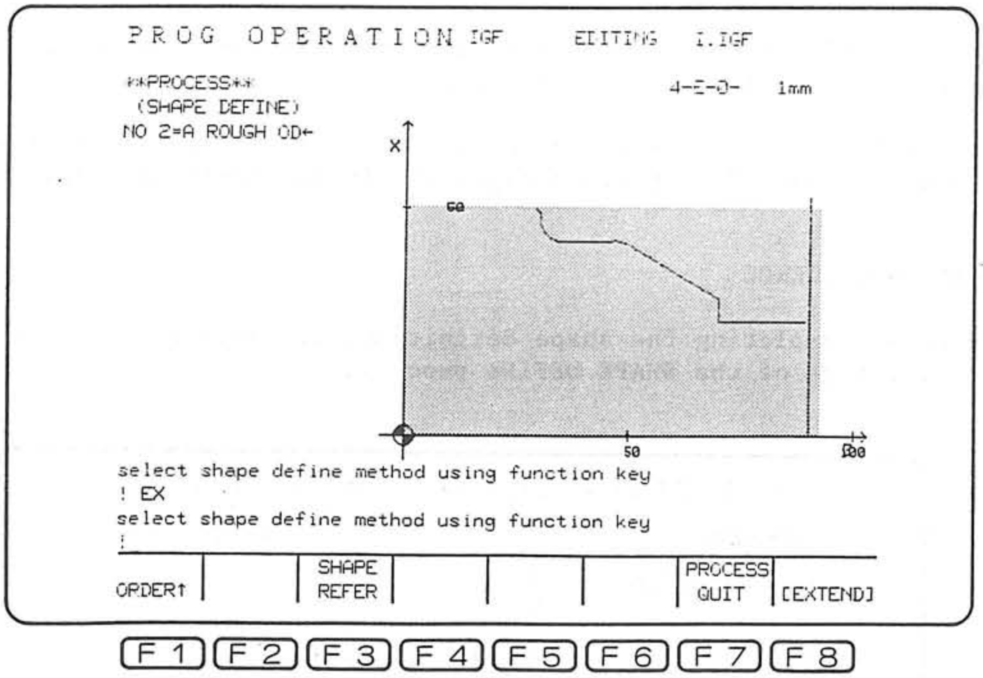
7-1. OPERATION SEQUENCE

(1) After completing the shape definition, the screen returns to the initial page of the SHAPE DEFINE process.



(2) Press the function key [F8] (EXTEND).

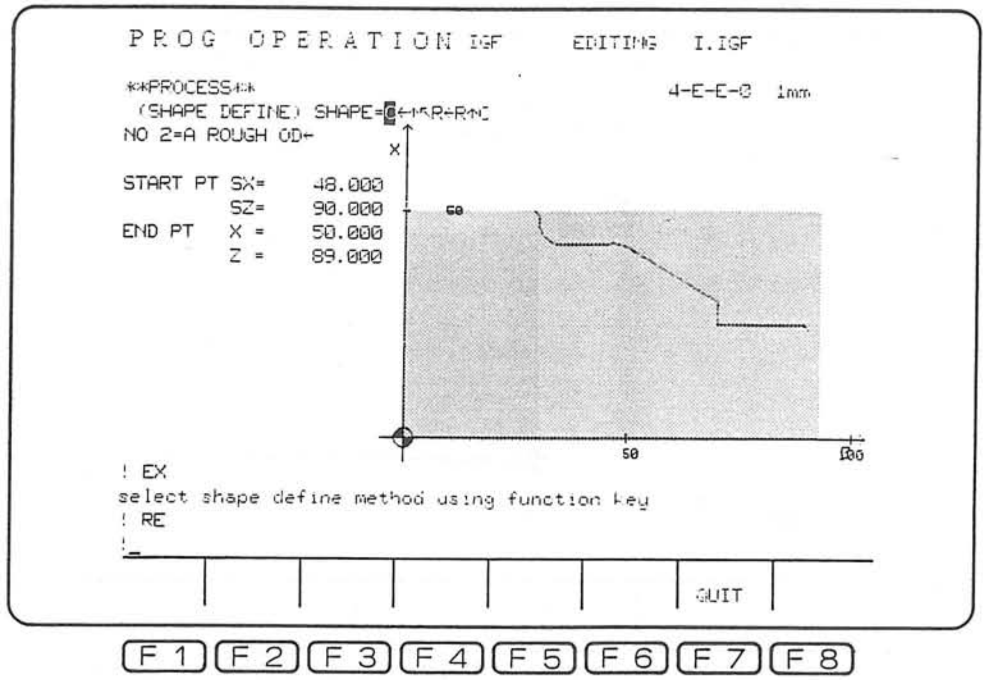
The set of function name guide display changes and the function name SHAPE REFER is displayed at function key [F3].



(3) Press the function key [F3] (SHAPE REFER).

The coordinates of the start and end points of the first shape (chamfer C) are displayed on the left of the screen.

In the shape drawing on the right of the screen, the end point is displayed by a flickering point.



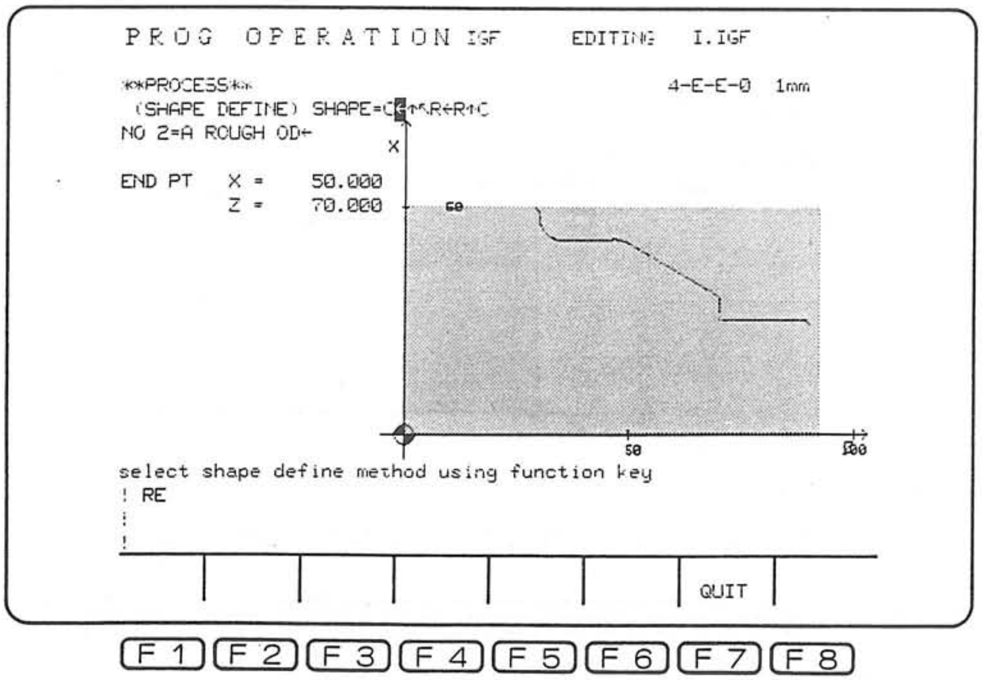
Shape elements are displayed in the upper area of the screen as "SHAPE =C←↑↖ R← R↑ C" and the cursor is located at the first "C". Here, SHAPE indicates the defined ROUGH OD shape, and the cursor indicates the shape whose end point is being indicated.

The procedure to display the shape end point coordinates following the C chamfer is explained below.

(4) Press the cursor control key [→] .

The cursor shifts to the next shape element "←", and the end point coordinates displayed on the CRT screen show the end of the cursor-indicated shape element ← .

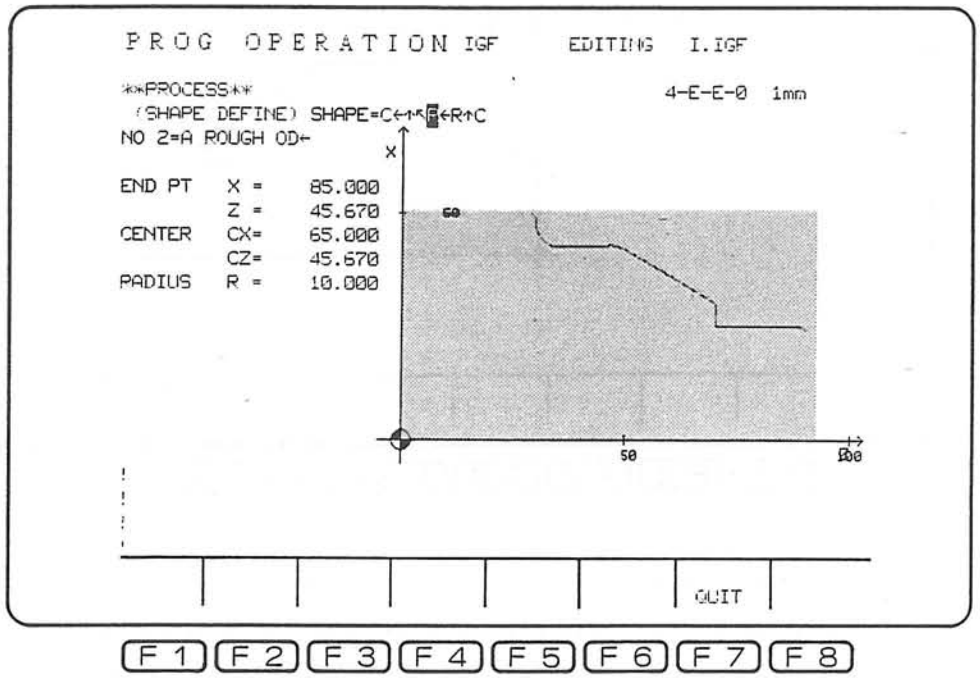
The flickering point in the drawn shape also shifts to the end point of the cursor-indicated shape element.



The procedure to display the end point coordinates of the rounding shape element R is shown below.

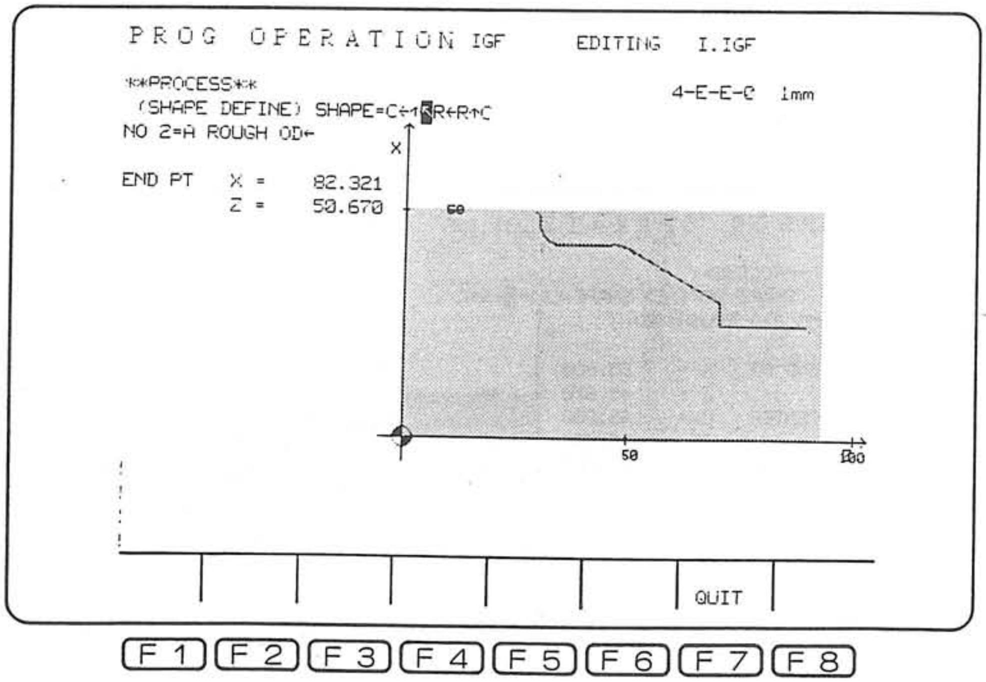
(5) Press the cursor control key [→] to locate the cursor at shape element "R".

For the arc shape, the screen displays the center coordinates and arc radius in addition to the end point coordinates of the defined shape.



The start point of the rounding arc is the end point of the preceding shape ↖. To display the end point of the preceding shape element, locate the cursor [←] at the preceding shape element.

(6) Move the cursor back to the shape element ↖ by pressing the cursor control key [←] .

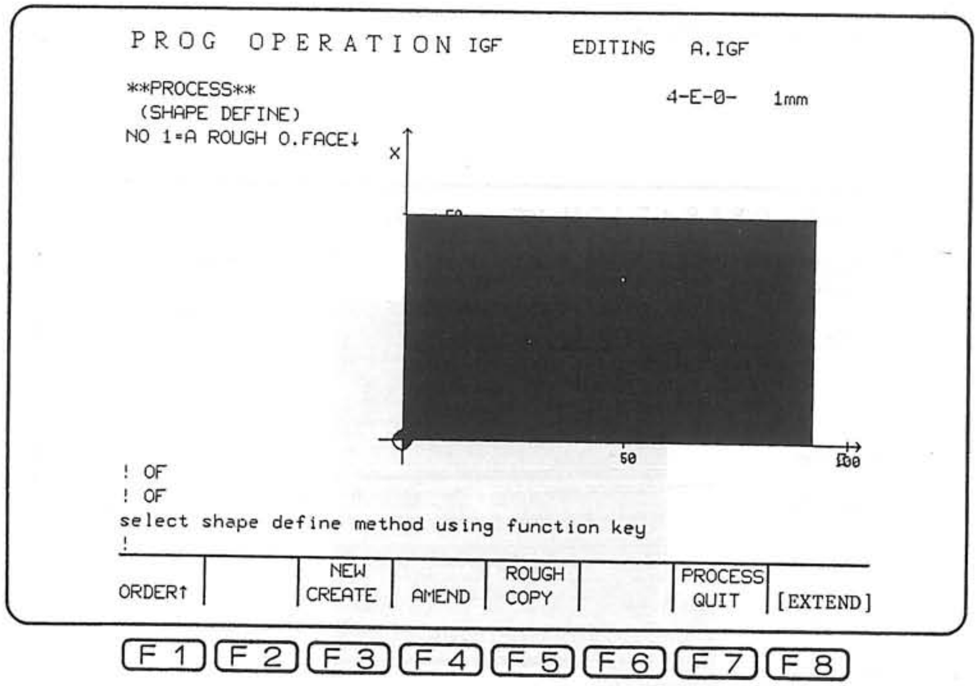


Using the same procedure, end point coordinates of each shape element are displayed on the screen. The cursor control keys [←] and [→] shifts the cursor forward and backward one by one each time it is pressed. When they are held down, the cursor moves continuously.

The cursor control key [←] is ignored when it is pressed if the cursor is located on the leftmost shape element. Similarly, the cursor control key [→] is ignored if pressed if the cursor is located on the rightmost shape element.

(7) To stop the shape reference function, press the function key [F7] (QUIT).

The first screen for the shape definition is displayed.



7-2. PRECAUTIONS

(1) The shape element reference function is effective for the following shape definition processes. If this function is called for other shape definition processes, an error occurs.

ROUGH
FINISH
COPY
FACE GENERAT

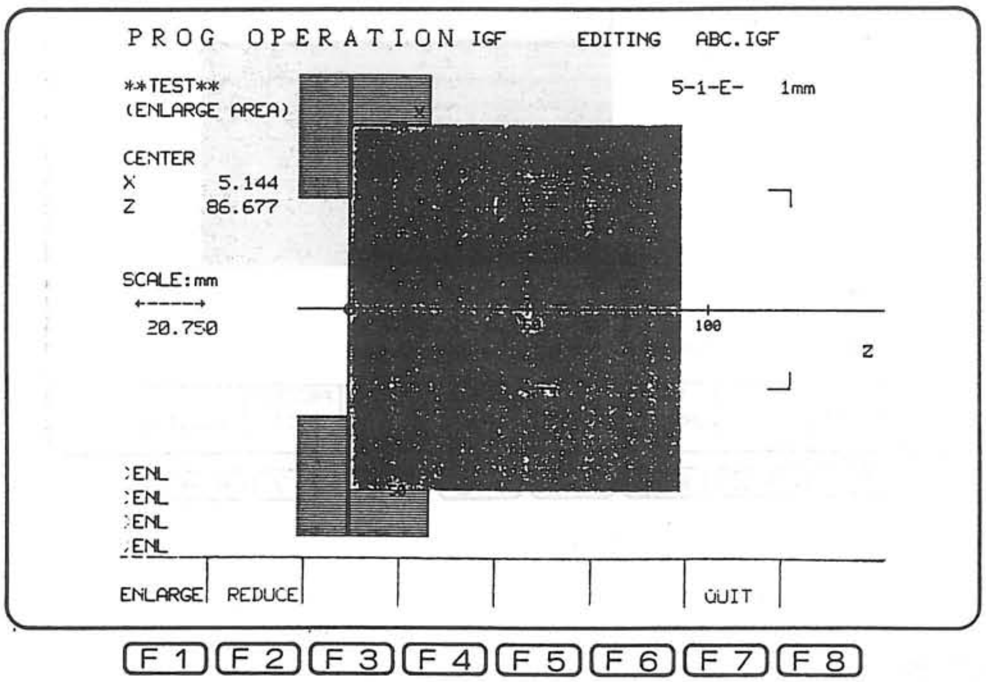
(2) Referencing a shape which has not been defined is impossible.

(3) The shape reference function is effective only for the active shape definition process. To reference the shapes defined in other processes, it is necessary to call this process first.

SECTION 8 PROGRAM TEST

The machining program made in the graphic edit operation is simulated on the CRT.

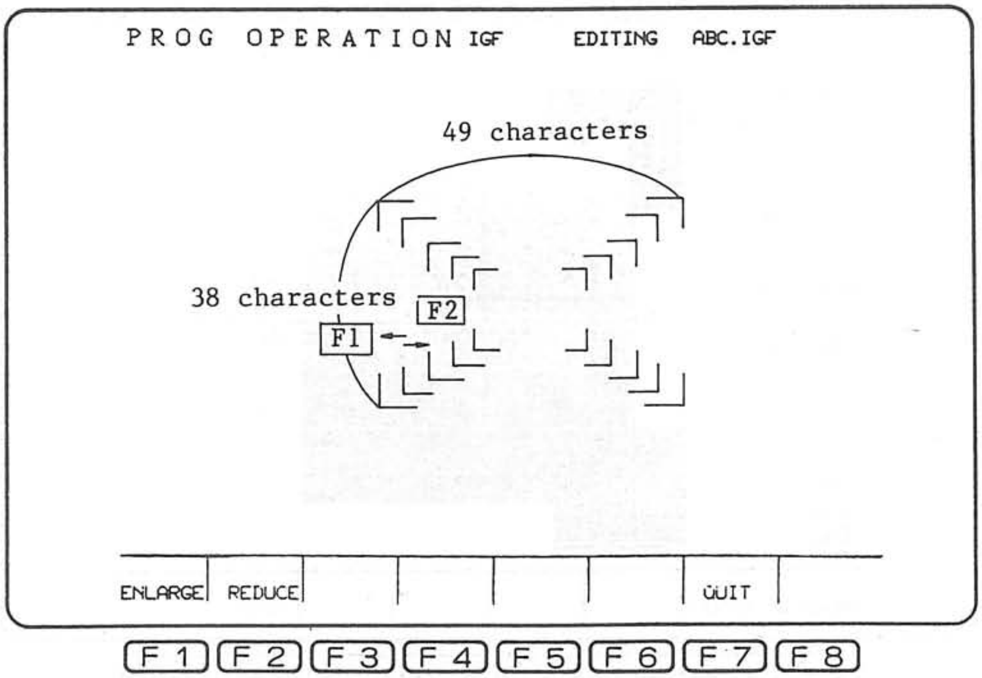
8-1. SCALE SETTING FOR SIMULATION



The area to be enlarged can be specified using the function keys [F1] (ENLARGE) and [F2] (REDUCE) in combination with the cursor keys

[↑] [↓] [←] [→] .

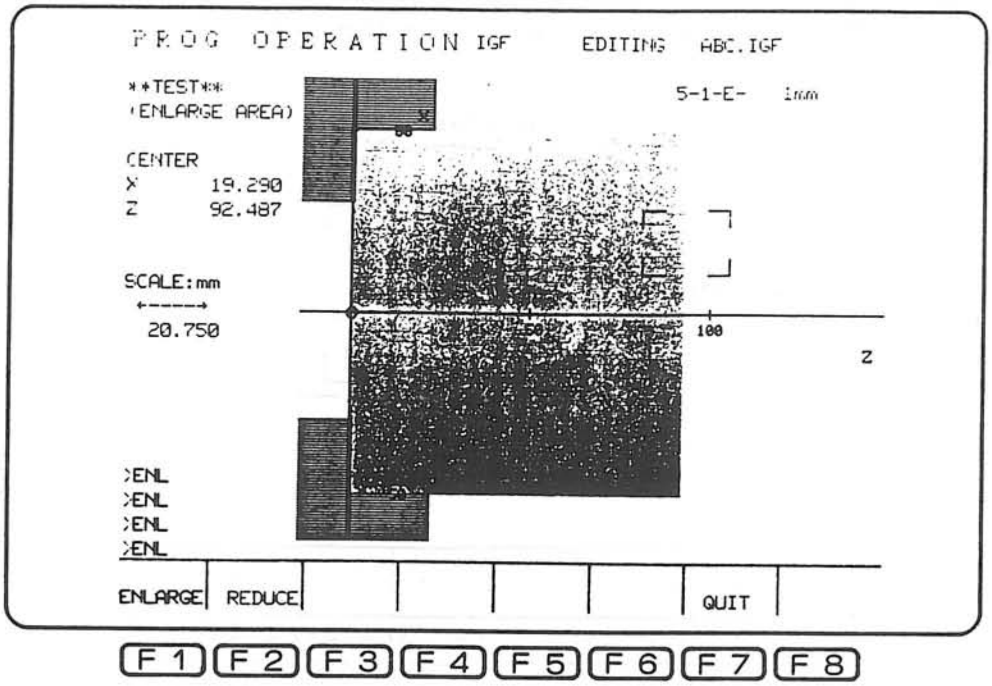
(1) Enlarging/reducing Frame



The frame size is set in the required size in 95 steps from the maximum area (38 characters x 49 characters) to the minimum area (1 character x 1.3 characters).

Pressing the function key [F2] (REDUCE) enlarges the frame area each time it is pressed. The function key [F1] (ENLARGE) reduces the frame area each time it is pressed.

(2) Frame Shift



The frame can be shifted within the maximum frame area. Pressing the cursor key moves the frame by a predetermined amount (*1) each time the key is pressed.

The CENTER X and Z data change in accordance with the frame shift.

*1: This amount is changeable by setting proper data at optional parameter (word) No. 49.

8-2. SETTING OPERATION MODE

The simulation operation mode includes the auto, block stop and process stop.

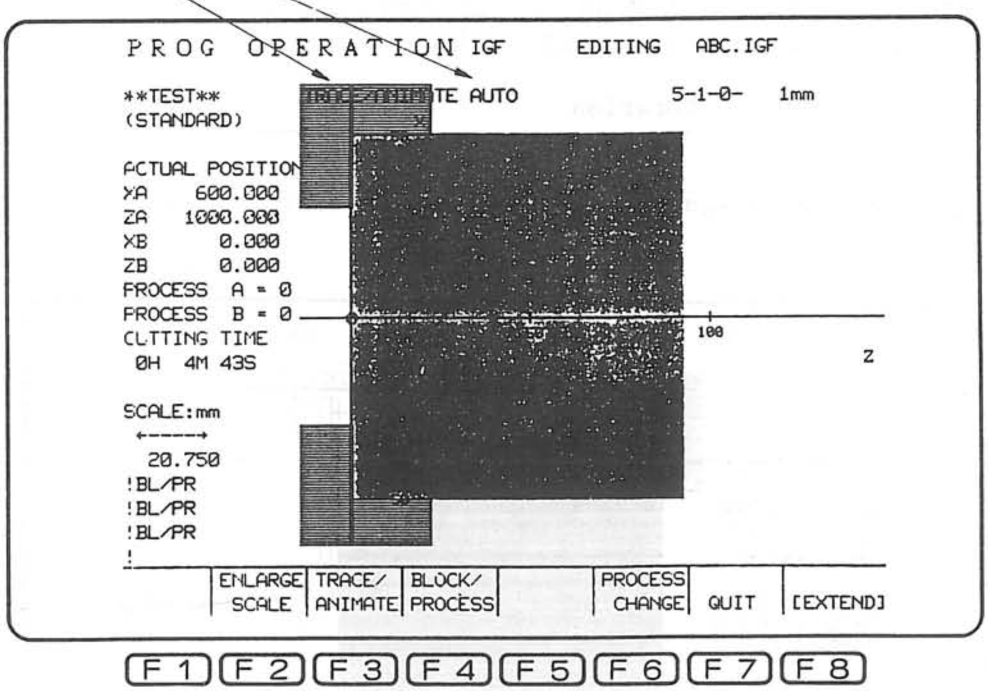
Pressing the function key [F4] (BLOCK/PROCESS) while the simulation is in halt changes the simulation mode in the following order sequentially.

AUTO MODE → BLOCK-STOP MODE → PROCESS-STOP MODE → (back to AUTO MODE)

Rotation

Display changes each time [F3] (TRACE/ANIMATE) is pressed.

Display changes each time [F4] (BLOCK/PROCESS) is pressed.



The selected operation mode is indicated on the CRT.

(1) Block-Stop Mode

The machining program made in the graphic edit operation is checked in units of blocks. Restart of the simulation is possible by pressing the function key [F1] (START).

(2) Process-Stop Mode

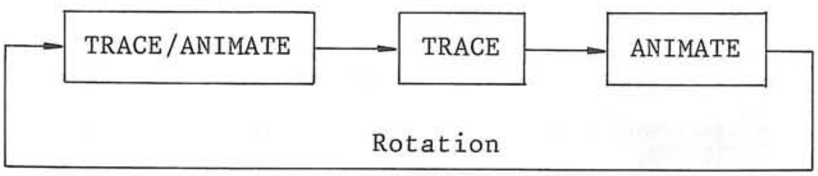
The machining program made in the graphic edit operation is checked in units of processes. Restart of the simulation is possible by pressing the function key [F1] (START).

(3) Auto Mode

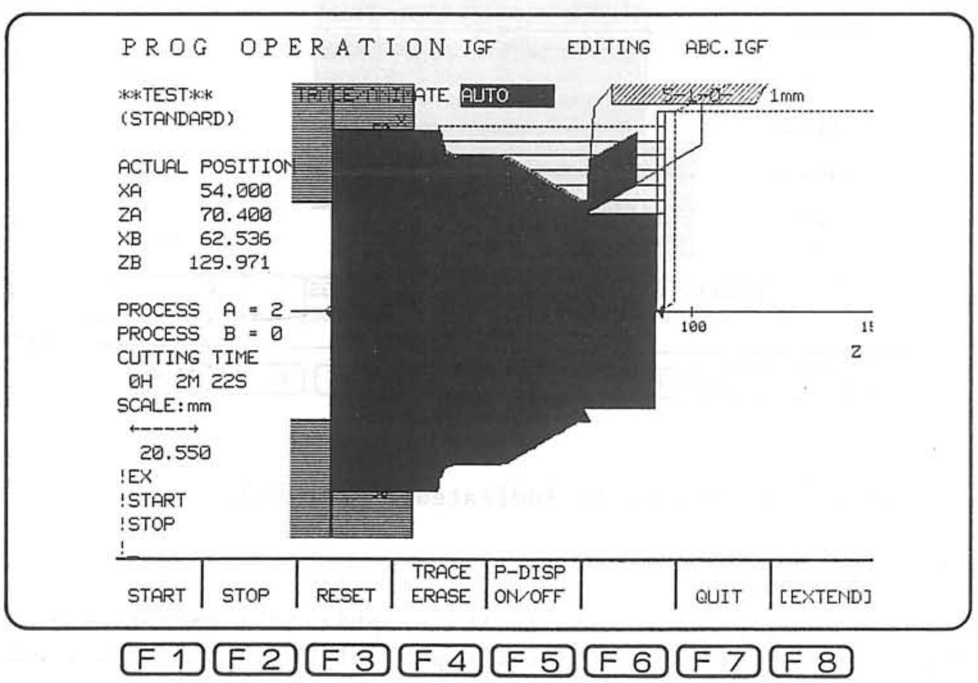
Pressing the function key [F1] (START) starts the simulation up to the end of the program.

8-3. SETTING GRAPHIC DISPLAY MODE

The display mode of the simulation operation includes the TRACE, ANIMATE and TRACE/ANIMATE modes. While the simulation is in halt, the graphic display mode is switched in the following sequence by pressing the function key [F3] (TRACE/ANIMATE).



The selected display mode is indicated on the CRT.



(1) Trace/Animate Mode

In the simulation operation, tool shape, chuck shape and work shape are displayed with the tool path displayed on the same display. The solid line indicates the cutting feed and broken lines indicate the rapid traverse. The cut portion is actually removed from the display.

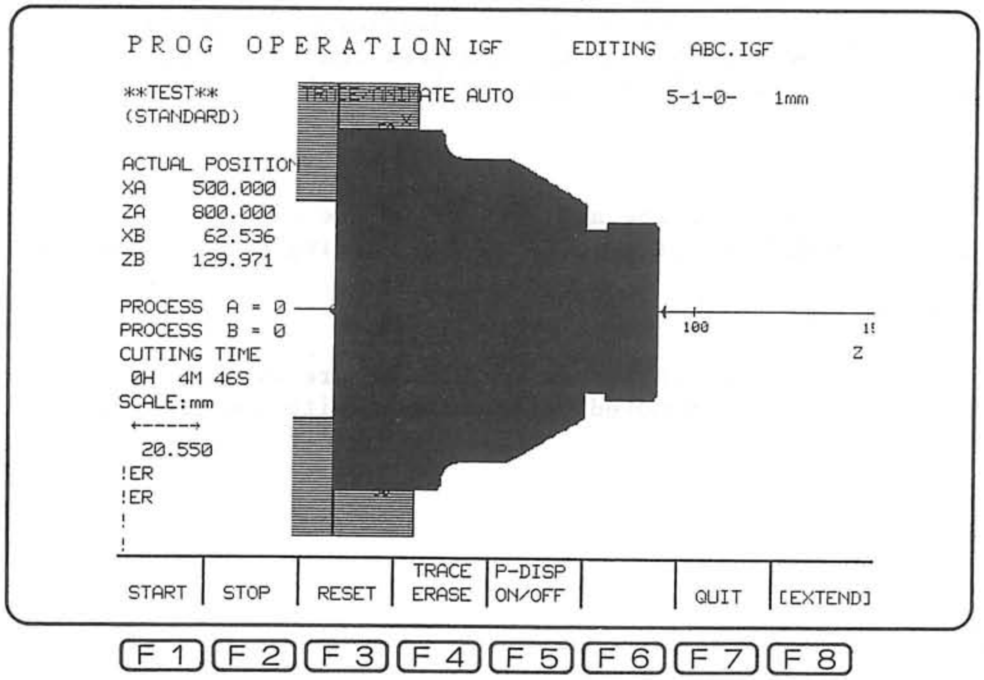
(2) Trace Mode

Tool path, chuck shape and work shape are displayed. In this display mode, removal of cut portion during cutting does not occur.

(3) Animate Mode

Tool shape, chuck shape and work shape are displayed. Cut portion of work is actually removed in accordance with the progress of cutting.

8-4. SIMULATION OPERATIONS



(1) Actual Position Display

The reference point of the cutting tool in reference to the program coordinate system is displayed. For the two-saddle model, coordinate values of the four axes, XA, ZA, XB and ZB, are displayed.

(2) Process Number Display

The process number of the process for which the simulation is being made is displayed. For two-saddle model, two processes for A- and B-turret are displayed.

(3) Cutting Time Display

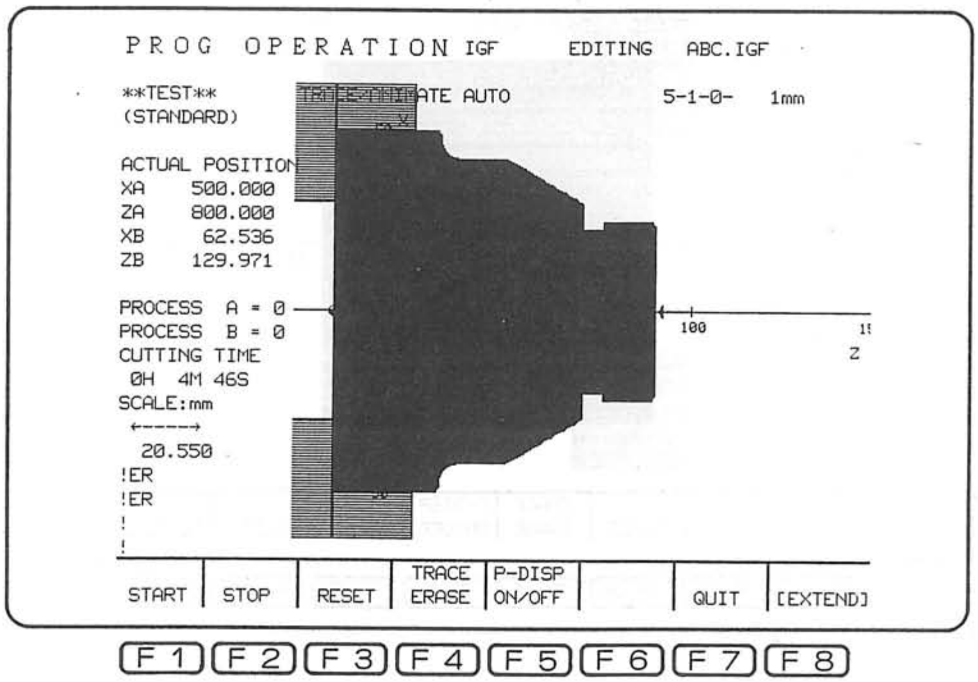
In the simulation operation, a total time of S, T and M function execution, cutting feed time and rapid traverse time is displayed at the end of the simulation.

(4) Start/Stop of Simulation

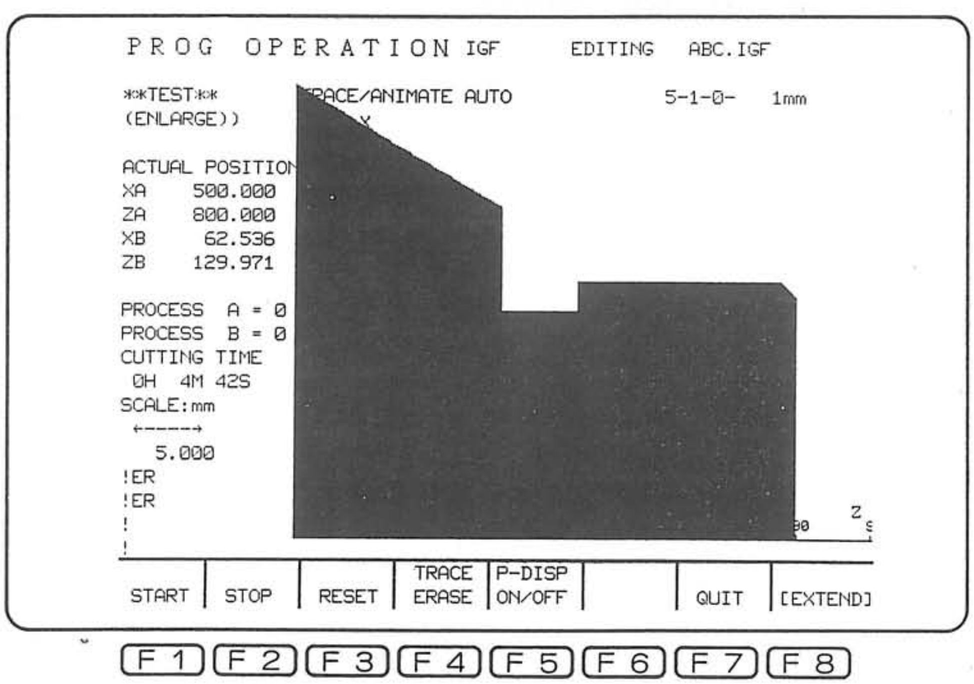
Pressing the function key [F1] (START) starts the simulation. When the function key [F2] (STOP) is pressed, the simulation is placed in the block-stop mode and the operation mode display is reversed.

(5) Switching between Standard and Enlarge Display

Pressing the function key [F1] (STANDARD/ENLARGE) switches the standard and enlarge display modes. Note that this change during the simulation is impossible. Enlarged display is possible when the ENLARGE SCALE has been set. If the ENLARGE SCALE is not set, selection of the enlarged display has no effect. The standard display size is based on the automatically set scale size.

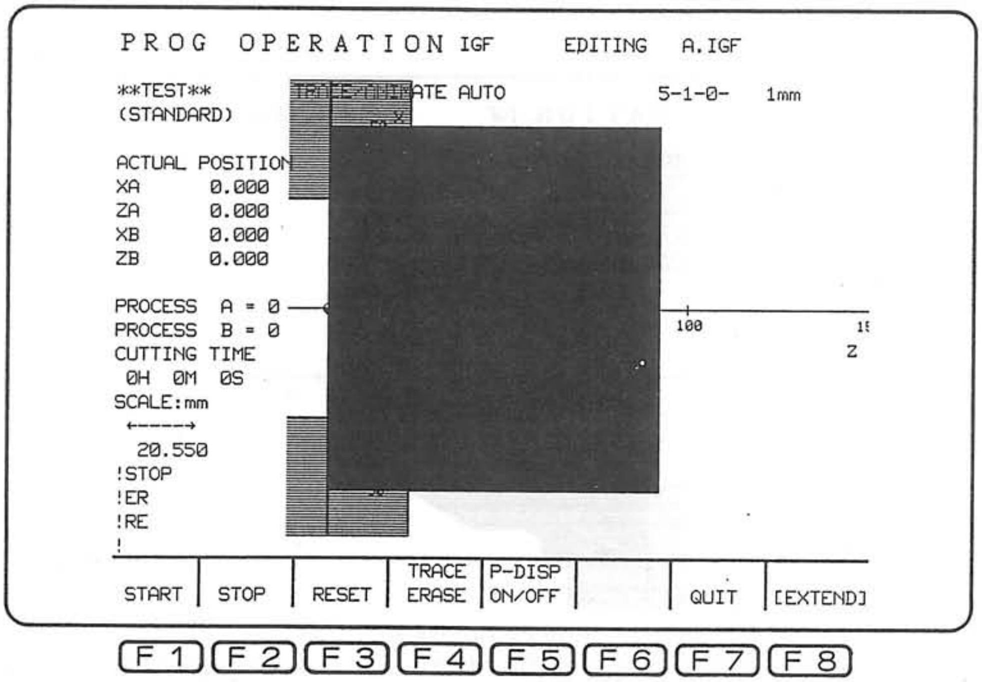


Page Key Operations



(6) Resetting Simulation

The function key [F3] (RESET) is used for resetting the simulation to the initial status with the blank shape displayed on the CRT. This key is effective only while the simulation is in halt.

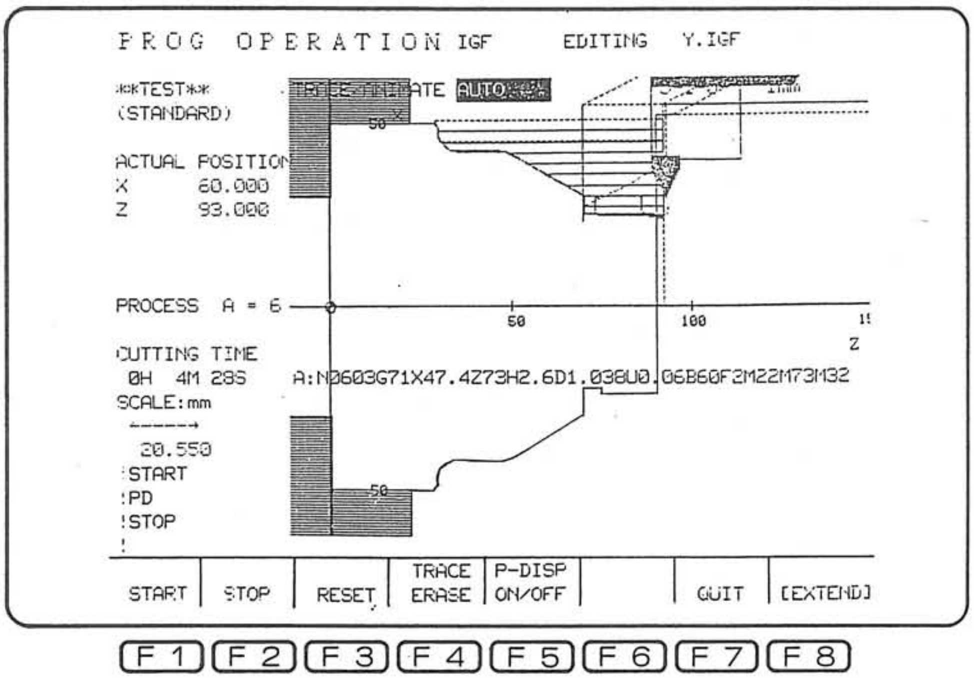


(7) Erasing Traces Displayed in Simulation Operation

The tool paths drawn during the simulation are erased by pressing the function key [F4] (TRACE ERASE).

(8) Program Display during Process Test

The block of the program currently executed during process test is displayed on the CRT screen. Display is turned on and off by the function key [F5] (P-DISP ON/OFF).



(9) Tool interference Check and Barrier Check

For the Two-saddle model, the tool interference check function for the tools on the A- and B-turret allows the NC program to be checked beforehand. The chuck barrier and tailstock barrier check functions are also used for checking the NC program..

Whether these check functions are to be activated or not are set by the IGF integer parameters.

If tool interference or crossing the barrier occurs during the simulation, the simulation stops instantaneously and the error message depending on the error type encountered is displayed.

The simulation can be resumed by pressing the function key [F1] (START).

Tool interference occurred:

Interference of tools on the A- and B-turrets

Tool entered chuck barrier:

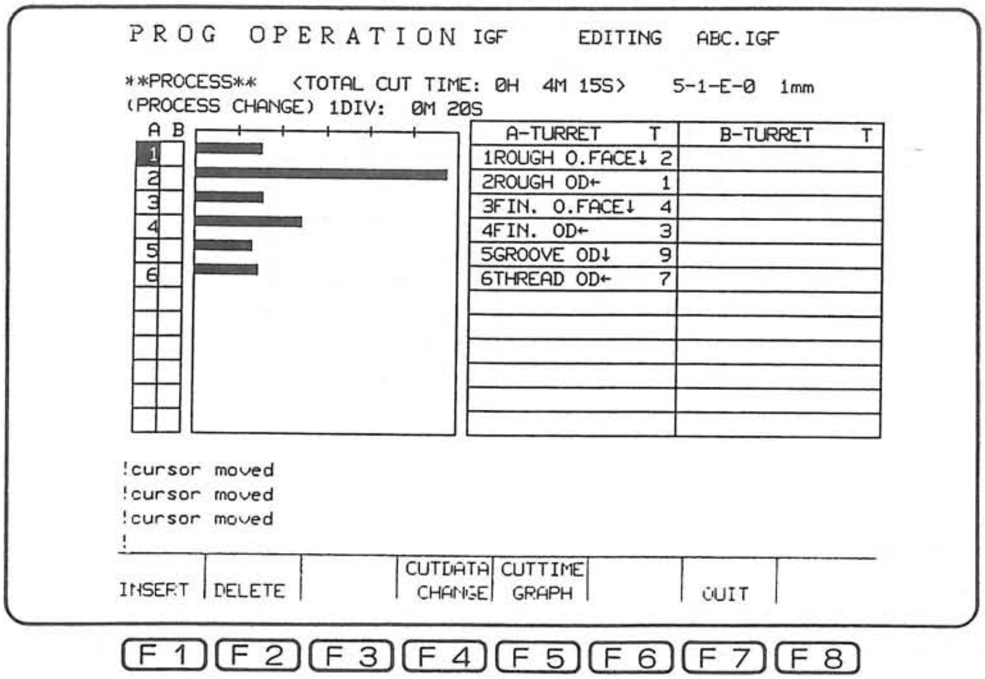
Tool point crossed the chuck barrier

Tool entered tailstock barrier:

Tool point crossed the tailstock barrier

SECTION 9 PROCESS CHANGE

Pressing function key [F6] (PROCESS CHANGE) enables the editing of the processes programmed, including replacing, insertion, deletion, continuous operation designation and change of cutting conditions.

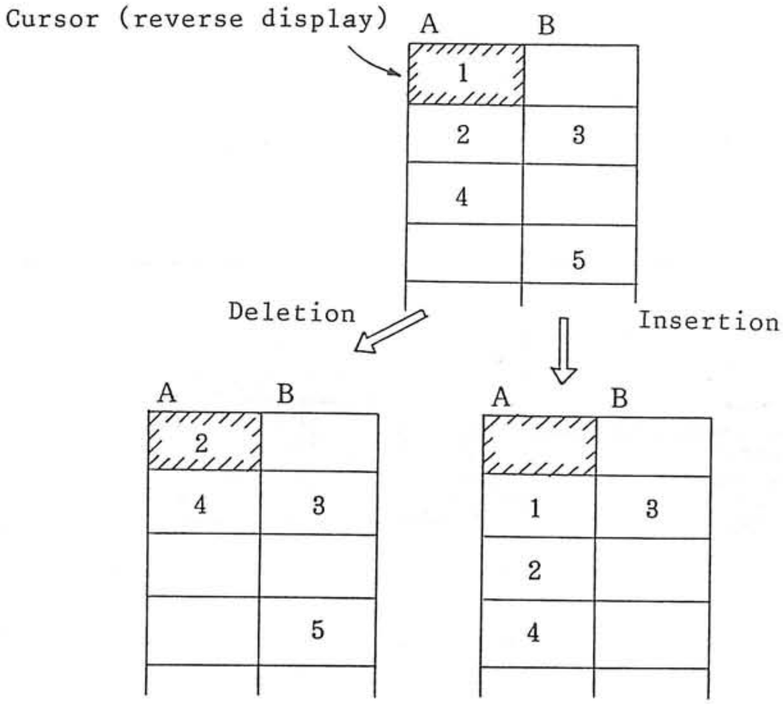


9-1. INSERTION/DELETION OF PROCESSES

The cursor keys [↑] [↓] [←] [→] on the operation panel move the cursor (reverse display) on the process table.

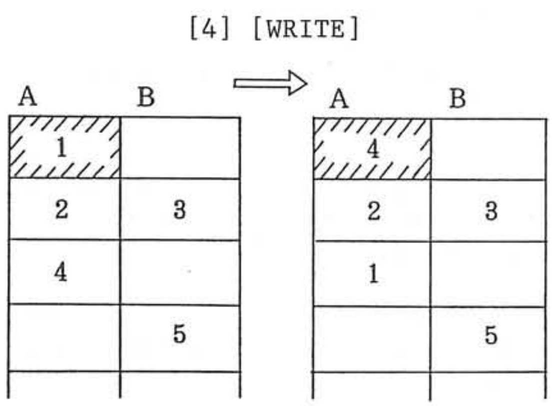
The function key [F1] (INSERT) is used for shifting the processes below the process on which the cursor is located downward by one. The process having been located by the cursor is left blank.

The function key [F2] (DELETE) is used for deleting the process located by the cursor. The processes indicated below the cursor located process are shifted upward by one.



9-2. REPLACING PROCESSES

First locate the cursor on the process which is to be replaced with the specified process. Then, key in the process for which the cursor located process is to be replaced. Pressing the WRITE key exchanges these two processes each other.



If a process number not present in the process table is specified, it will cause an error (process number designate).

9-3. DESIGNATION OF CONTINUOUS EXECUTION OF PROCESS

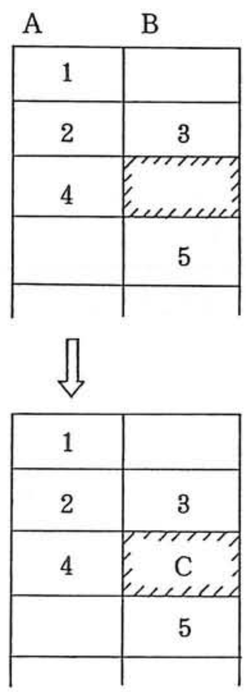
Leave the process just below the one to be executed continuously and enter "C" in the blank process by pressing the C and WRITE keys.

For instance, if the "3" process of the B-turret is to be executed continuously in the simultaneous four-axis control mode with the "2" and "4" processes of the A-turret, this continuous execution mode can be designated in the following operations.

1) Locate the cursor at the process under "3" process of B-turret.

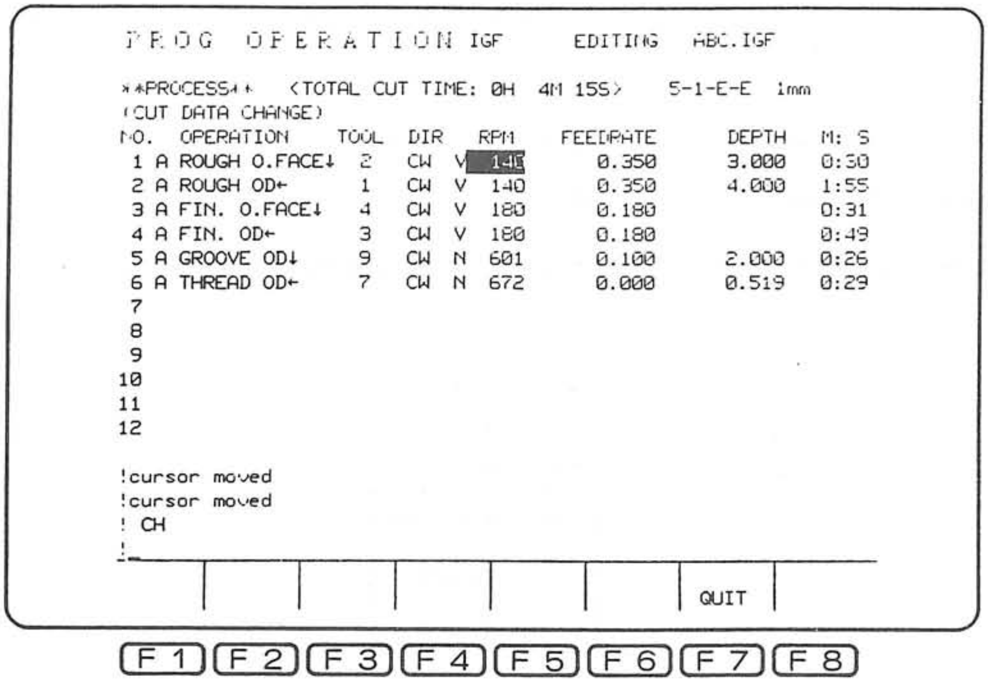
2) Press the keys C and WRITE.

3) The character "C" will appear in the process table.



9-4. CHANGING CUTTING CONDITIONS

Pressing function Key [F4] (CUT DATA CHANGE) from CUT DATA CHANGE table, it is possible to change the spindle speed (RPM), feedrate (FEEDRATE) and depth of cut (DEPTH).



SECTION 1Ø MAKING NC PROGRAMS

The spindle control commands, coolant on/off control commands and tool change (turret index) commands are generated in the stage where the IGF data is compiled into NC programs.

1Ø-1. SPINDLE CONTROL

(1) Spindle Rotation Direction Control

a) The spindle rotation direction is controlled in accordance with the commands specified in each process.

b) If the spindle rotation direction is to be changed, the spindle is stopped once before the rotation direction is changed.

c) Between the processes, the spindle keeps rotating unless its rotation direction is changed between the two consecutive processes.

Note: For the control equipped with the mirror image specification, the spindle is stopped once before the spindle rotation direction is changed if the actual spindle rotation direction is changed.

(2) Spindle Speed Control

Constant Cutting Speed Mode ON:

a) The spindle is started at the constant speed mode. The spindle rpm at the cutting start point is used for rotating the spindle.

b) After the completion of the positioning at the reference point, the constant cutting speed mode is activated.

c) After the completion of cutting, the spindle rotates at the speed selected in step a).

Constant Cutting Speed Mode OFF:

a) The spindle speed is determined based on the average cutting diameter. For drilling operation, the average cutting diameter is equal to the drill diameter. Usually, the average cutting diameter will be the average of the maximum and the minimum cutting diameters.

Note: Whether the constant cutting speed mode is selected or not is set by the IGF parameter. For drilling and thread cutting operations, however, the setting of this parameter is ignored and these operations are not executed in the constant cutting speed mode even if so set.

(3) Spindle Speed Range Control

a) The spindle speed range is automatically determined from the spindle rpm which is calculated based on the average cutting diameter.

b) If more than one spindle speed range is selectable, the lowest range is selected.

c) If the IGF parameter setting is improper*, such a spindle speed range is ignored and selection to such a range is not made.

* This indicates the setting of max. rpm $\leq$ min. rpm in a range.

1Ø-2. COOLANT ON/OFF CONTROL

(1) The coolant on/off command can be programmed for each process.

(2) The coolant supply stops after the completion of each process.

1Ø-3. TOOL CHANGE CONTROL

(1) The tool change control refers to the control in which the air cutting cycle is controlled and the T code is controlled.

(2) The T code control is accomplished by the tool number for selecting the tool to be used and the tool offset number.

(3) The air cutting cycle control includes the following operations:

1) Retraction to the initial turret index position

2) Approach to the cycle reference point

3) Retraction to the clearance boundary at the completion of cutting

4) Retraction to the turret index position from the clearance boundary

5) Movements to the turret index position for the next process

6) Movements to the cycle reference point of the next process when turret index is not necessary

7) Final retraction

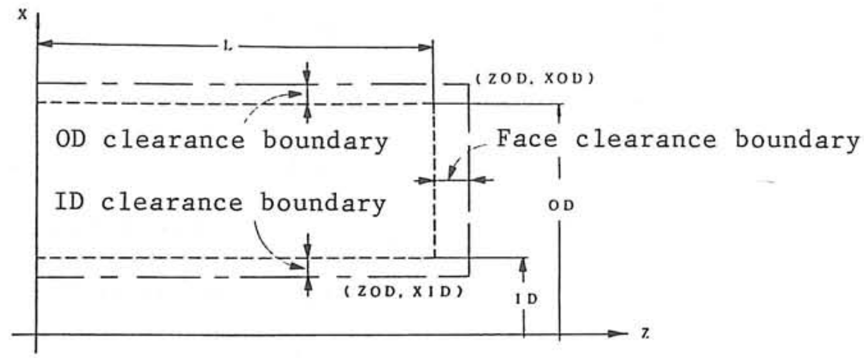
(4) Three types of data indicated below are necessary for the air cutting cycle control:

a) Clearance
b) Classification of turret index position
c) Classification for cutting direction for air cutting

a) Clearance boundary

- OD clearance boundary ...... XOD
- ID clearance boundary ...... XID
- Face clearance boundary .... ZOD

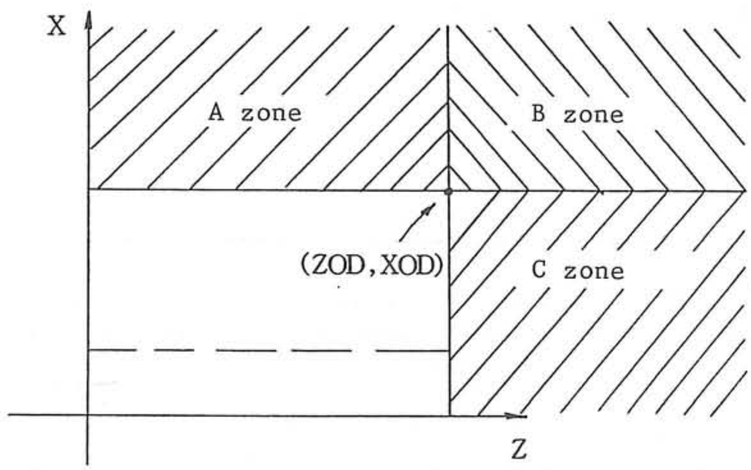
These indicate the boundary shown in alternate short and long dashes line in the figure below.



Note: After the completion of drilling, the ID clearance boundary is shifted by the amount equivalent to drill diameter.

b) Classification of turret index position

The turret index position is classified into three zones as illustrated below by separating by the clearance boundary.



A zone:

Turret index position ZT ≦ Face clearance boundary ZOD
Turret index position XT > OD clearance boundary XOD

B zone:

Turret index position ZT > Face clearance boundary ZOD
Turret index position XT > OD clearance boundary XOD

C zone:

Turret index position ZT > Face clearance boundary ZOD
Turret index position XT ≦ OD clearance boundary XOD

c) Classification for cutting direction for air-cutting

Every cutting process has the data concerning the cutting process kind and cutting direction. The direction of air-cutting is classified depending on the combinations of these two types of data.

There are four air-cutting directions as indicated below. Typical process kinds and direction of cuts included in each air-cutting direction are listed in the right column.

OD	OD ROUGH, THREAD OD, GROOVE OD ↓ , RECESS OD
OD FACE	OD FACE ↓ , THREAD FACE ↓ , GROOVE FACE←
ID FACE	ID FACE ↑ , THREAD FACE ↑ , DRILL (all)←
ID	ID ROUGH, THREAD ID, GROOVE ID ↑ , RECESS ID

1) Retraction to the Initial Turret Index Position

a) Before the cutting process programmed executed, the turret is retracted to the turret index position for the first cutting process. This movements are accomplished in the simultaneous two axis control mode.

b) On the two-saddle model, retraction to the turret index position is performed in the order from the A turret to the B turret.

c) For the mirror image function, retraction is carried out to the turret index position specified in the first cutting process.

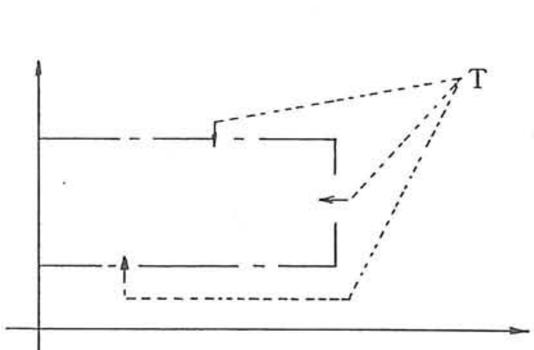
2) Approach to the Cycle Reference Point

a) The T command is output in accordance with the tool number and the offset number specified by the tool which is designated in the cutting process.

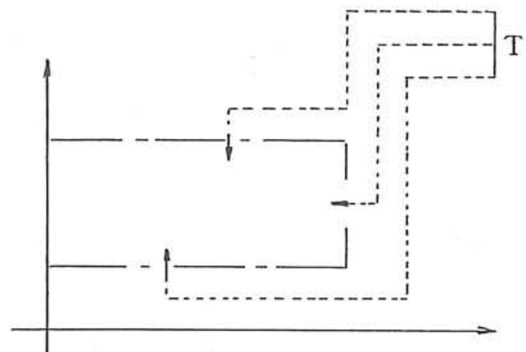
b) The approach path consists of the movements up to the clearance boundary and the movements from the clearance boundary to the reference point. It is determined based on the classification of the turret index position and the classification of the air-cutting direction.

c) The movements from the turret index position to the clearance boundary can be made in the simultaneous two axis control mode or the simultaneous single axis control mode depending on the setting of the IGF parameter.

Approach from the B Zone

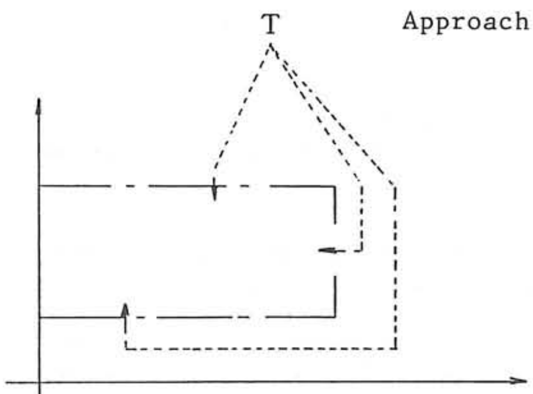


Simultaneous two axis movements

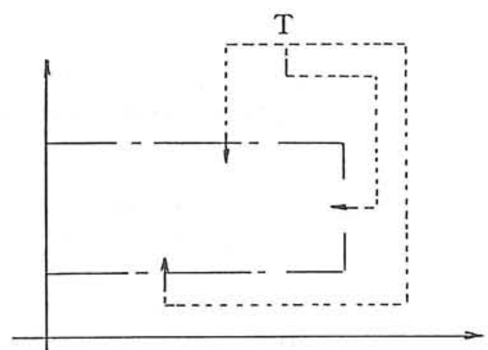


Simultaneous single axis movements

Approach from the A Zone

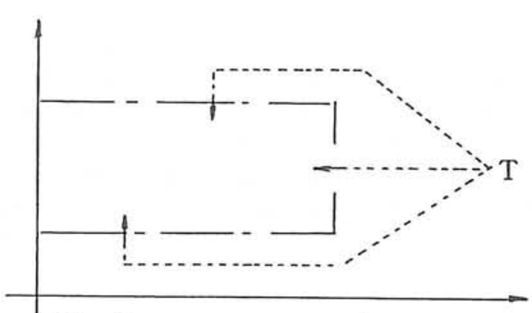


Simultaneous two axis movements

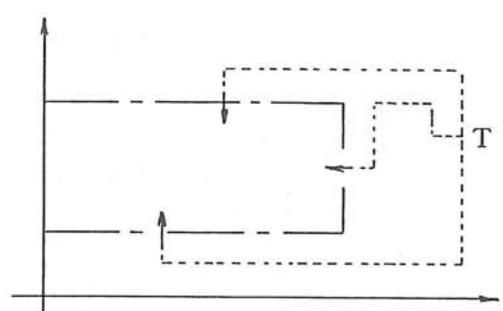


Simultaneous single axis movements

Approach from the C Zone



Simultaneous two axis movements

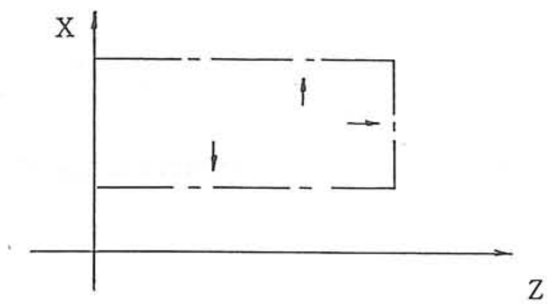


Simultaneous single axis movements

3) Retraction to the Clearance Boundary at the Completion of Cutting

a) After the completion of cutting, the cutting tool is retracted up to the clearance boundary in accordance with the air-cutting direction.

b) Retraction is made only by one axis.



4) Retraction to the Turret Index Position from the Clearance Boundary

a) When the tool to be used in the next process differs from the tool having been used, the turret is further retracted up to the turret index position from the clearance boundary.

b) In the retraction to the turret index position, tool offset amount is not taken into consideration (T**ØØ).

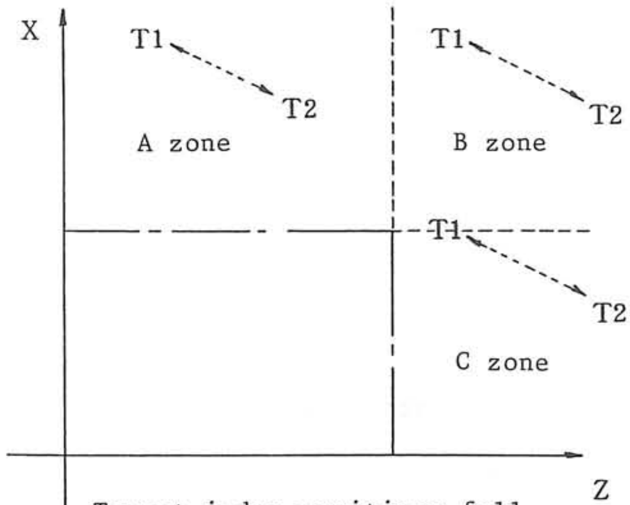
c) The retraction path is determined depending on the turret index position classification and the air-cutting direction classification.

d) Actually, the path reverse to the approach path to the reference point indicated in 2) is generated.

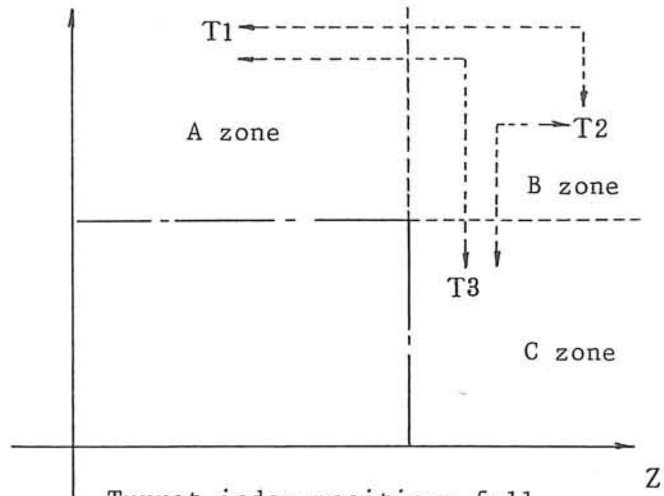
5) Movements to the Turret Index Position for the Next Process

a) If the turret index position selected for the next process is different from the turret index position for the present process, the turret is moved to the new turret index position.

b) The tool path is determined based on the classification of the turret index positions between the present and the next processes. That is, if the turret index positions fall within the same classification, movements to the new turret index position are carried out in the simultaneous two axis movements. If they fall within different zones, then axis movements occur in parallel with the coordinate axes in the simultaneous one axis movements.



Turret index positions fall in the same zone



Turret index positions fall in different zones

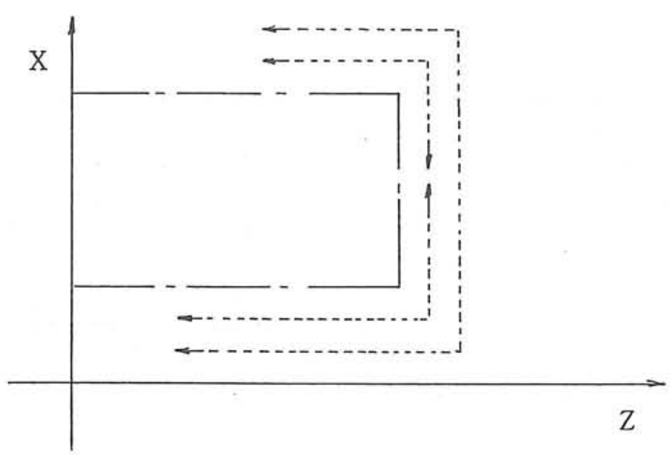
6) Movements to the Cycle Reference Point of the Next Process when Turret Index is not Necessary

a) If the same tool as used in the present process is used in the next process, the turret moves to the new reference point.

b) Tool path is determined from the present air cutting direction and that of the next process.

c) The movements to the clearance boundary are carried out in the single axis control mode. See the illustration below.

d) Tool offset may be switched during axis movements.

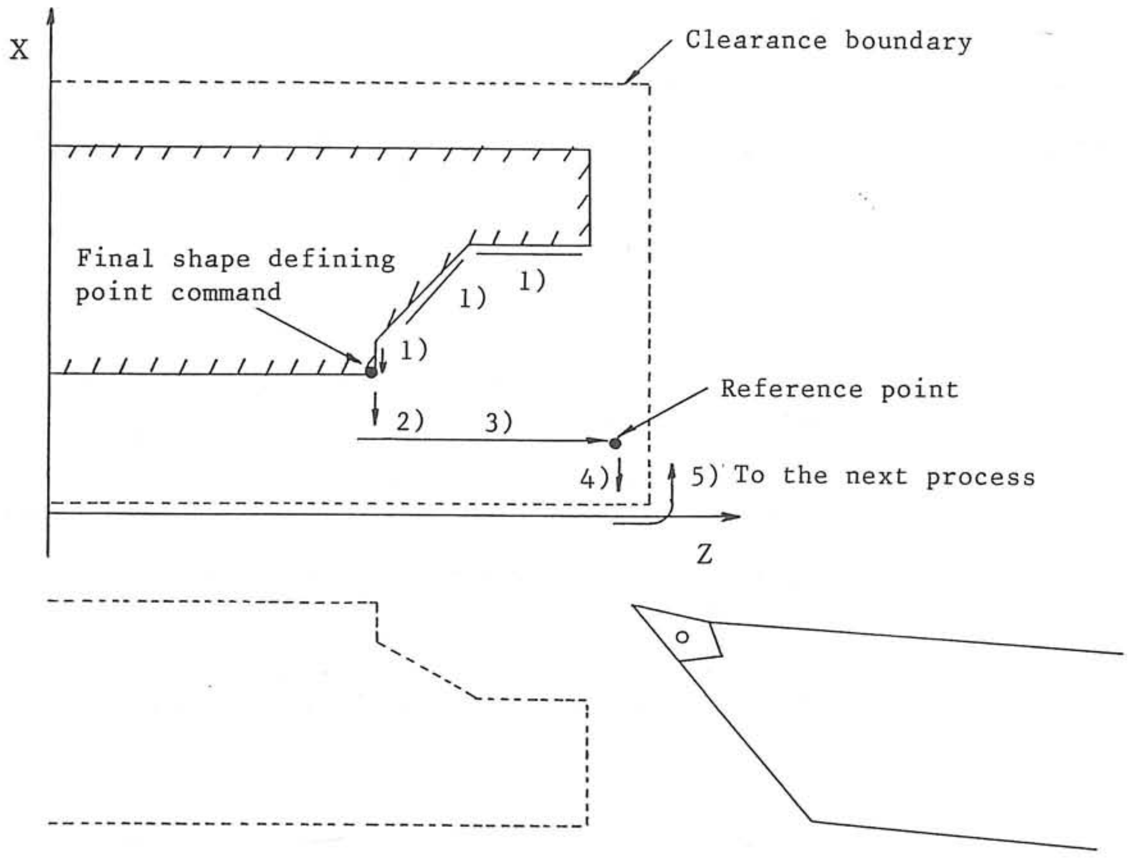


7) Final Retraction

a) After the completion of cutting, the tool is retracted up to the turret index position for the final process.

b) The retraction path is the reverse path to the approach up to the reference position explained in 2).

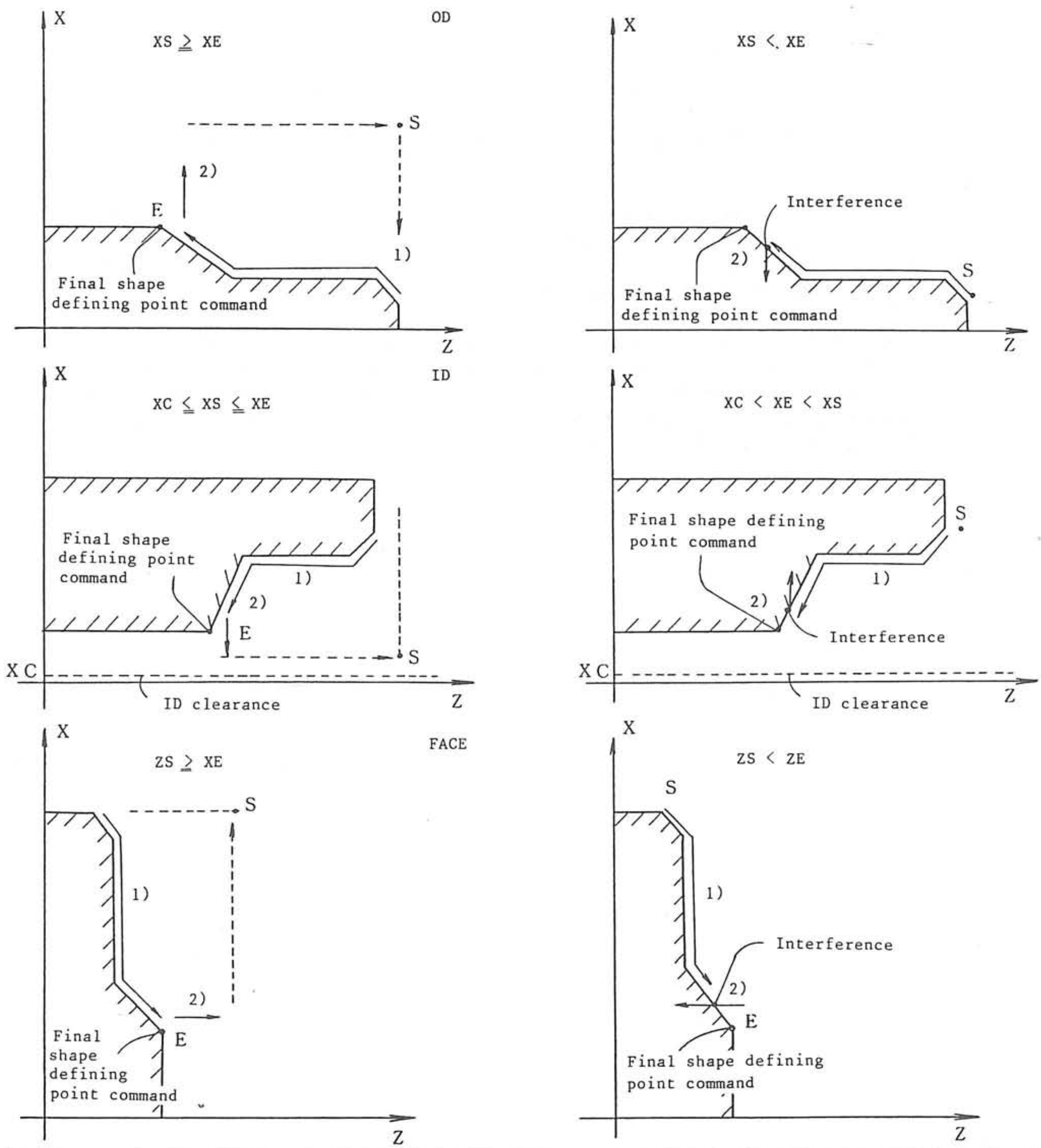
Example: Retraction of Turret after the Completion of Finishing Cycle



The cutting tool used in the finishing cycle is retracted along the path from the final shape defining point, reference point and up to the clearance boundary. (In the rough, copy, thread cutting, drilling, grooving, and recessing, the cutting tool is also retracted from the end point to the clearance boundary via the reference point.)

In the multi-lead thread cutting or the multi-grooving, retraction to the clearance boundary occurs from the start point of the finally shifted thread lead or groove.

Warning: Retraction mode from the final shape defining point to the reference point differs depending on the direction of cuts selected. That is, when the cutting is carried out in the longitudinal direction, it is returned in the order of X and Z axes. If it is carried out in the transverse direction (face cutting), retraction to the reference point is made in the order of Z and X axes. In either case, retraction is carried out only in the simultaneous single axis control mode. Therefore, the reference point must be established carefully if the final shape defining point is taken at the points indicated below.



SECTION 11 2-SPINDLE PROGRAM

The following is the basis for the ways of use of IGF for making NC programs for twin-spindle specification.

(1) Define cutting shape or cutting conditions through the use of IGF graphic edit function as usual.

(2) When making an NC program, NC program output from the IGF is made as subprogram. (Note that turret designation codes G13, G14 or synchronization code P will not be output.)

Example:

When giving a program name (file name), input a name.

! W1.SSB OØØØ1* [WRITE] ..... The program is made as a system subprogram.

* The name given here (OØØØ1) is the subprogram name which is used to call out the program from the main program. (Refer to the Instruction Manual, Section 8-7, "Making NC Program" as to how to make a subprogram.)

(3) Make a main program when actually cutting with the subprogram made through the IGF operations (through the edit function). When making a main program, designate whether the part program made through the IGF operations is to be executed with either A- or B-turret using G13 (A-turret) and G14 (B-turret).

Example:

A.MIN

O1ØØ	Designate program name.
NØØØ1 G13	Designate A-turret with G13. Until G14 is designated, A-turret is used for cutting.
NØØØ2 CALL OØØØ1 P1Ø	Designate the subprogram which is to be executed on A-turret. (Refer to "*" of (2).) Designate P1Ø for synchronization with B-turret.
NØØØ3 G14	Designate B-turret with G14.
NØØØ4 CALL OØØØ2 P1Ø	Designate the subprogram which is to be executed on B-turret. (In this case, a part program which is made in the similar way as that in (2) but different from the program designated for A-turret is designated.) Designate P1Ø for synchronization with A-turret.
NØØØ5 MØ2	Designate MØ2 at the end of program.

In this example, different part programs are designated for A-and B-turret. However, there is no problem if the same part program is called out for A- and B-turret in order to cut the same shape. In addition, the program may be executed only on one turret.

Even when the program contains both G13 and G14, individual cutting with A- or B-turret is possible with the turret selection switch on the operation panel.

The IGF can be taken advantage of even better by making this kind of main program and using it for A/B separate operation, A/B simultaneous 4-axis cutting and with the NC loader.

(4) Carry out cutting by selecting program in the conventional automatic operation as shown below.

Example 1

=PS [WRITE] (=PS A.MIN.O1ØØ,W1.SSB [WRITE])

Since the method of use indicated above is used as the basis with the twin-spindle specification, when making of main program is attemped through the IGF, no question concerning which turret to be used (A or B) is posed. Therefore, there will be no turret designation codes G13 or G14 or synchronization code P are output to NC programs which are made through the IGF.

Example 2

a) Make a subprogram using the IGF.

! X1.SUB OØØØ3 [WRITE]

Note: It is possible to make this subprogram as the system subprogram by assigning X.SSB.

b) Make a main program by editing.

B.MIN

```
O3ØØ
NØ3Ø1  G13
NØ3Ø2  CALL  OØØØ3  P1Ø
NØ3Ø3  G14
NØ3Ø4  CALL  OØØØ3  P1Ø
NØ3Ø5  MØ2
```

c) In the automatic operation, select the program as shown below and carry out machining.

=PS B.MIN.O3ØØ,X1.SUB [WRITE]

- X1.SUB: Designate the file name which contains the subprogram which is called in the main program. (SSB: Omissible when system subprogram is used.)
- O3ØØ: Designate the main program name. (Omissible when this program exists at the beginning of file.)
- B.MIN: Designate the file name which contains the main program. (Omissible when A.MIN is used.)

SECTION 12 IGF FOR PRECISION LATHES

Differences between the IGF for conventional NC lathes and precision NC lathes are indicated below.

12-1. UNIT SYSTEM

Unit systems of Ø.1 μm and Ø.ØØØØ1 inches are added. See the setting of parameter (bit) data.

Setting of data at bit Ø, 1 and 2 of parameter (bit) No. 3	Unit System
ØØØ	1 μm
ØØ1	Ø.ØØØ1 inch
Ø1Ø	1 mm
Ø11	1 inch
1ØØ	1Ø μm
1Ø1	Ø.ØØØØ1 inch
11Ø	Ø.1 μm

Unit amount for each unit system setting is indicated below (excluding material data).

Dimension Parameter	Ø.1 μm	1 μm	1Ø μm	1 mm	Ø.ØØØØ1 inch	Ø.ØØØ1 inch	1 inch
Length	Ø.ØØØ1 (mm)	Ø.ØØ1 (mm)	Ø.Ø1 (mm)	1 (inch)	Ø.ØØØØ1 (inch)	Ø.ØØØ1 (inch)	1 (inch)
Feed per Rev.	Ø.ØØØ1 (mm/rev)	Ø.ØØ1 (mm/rev)	Ø.Ø1 (mm/rev)	1 (mm/rev)	Ø.ØØØØ1 (inch/rev)	Ø.ØØØ1 (inch/rev)	1 (inch/rev)
Angle (°)	Ø.ØØØ1	Ø.ØØ1	Ø.Ø1	1	Ø.ØØØØ1	Ø.ØØØ1	1
Speed (rpm)	1	1	1	1	1	1	1
Surface Speed (Cutting Speed)	1 (m/min)	1 (m/min)	1 (m/min)	1 (m/min)	1 (feet/min)	1 (feet/min)	1 (feet/min)

For the precision lathes, length, feed and angle data can be set one place more to the right to the decimal point than the IGF for the conventional NC lathes.

12-2. UPPER AND LOWER LIMITS OF DATA SETTING FOR GRAPHIC EDIT, TOOL DATA SET AND PARAMETER SET (DIMENSION PARAMETER)

Dimension Parameter	Other than Precision Lathe Spec.				Precision Lathe Spec.			
	Metric System		Inch System		Metric System		Inch System	
	Minimum Value	Maximum Value	Minimum Value	Maximum Value	Minimum Value	Maximum Value	Minimum Value	Maximum Value
Length *1, *3	-99999.999 (mm)	+99999.999 (mm)	-3937.ØØ78 (inch)	+3937.ØØ78 (inch)	-9999.9999 (mm)	+9999.9999 (mm)	-393.7ØØ78 (inch)	+393.7ØØ78 (inch)
Feed per Rev.	Ø.ØØ1 (mm/rev)	+99999.999 (mm/rev)	Ø.ØØØ1 (inch/rev)	+3937.ØØ78 (inch/rev)	Ø.ØØØ1 (mm/rev)	+9999.9999 (mm/rev)	Ø.ØØØØ1 (inch/rev)	+393.7ØØ78 (inch/rev)
Angle (°) *2	-36Ø.ØØØ	+36Ø.ØØØ	-36Ø.ØØØØ	+36Ø.ØØØØ	-36Ø.ØØØØ	+36Ø.ØØØØ	-36Ø.ØØØØØ	+36Ø.ØØØØØ
Speed (rpm)	1	9999	1	9999	1	9999	1	9999
Surface Speed (Cutting Speed)	1 (m/min)	3ØØØ (m/min)	1 (feet/min)	9999 (feet/min)	1 (m/min)	3ØØØ (m/min)	1 (feet/min)	9999 (feet/min)

*1 For DEPTH and ALLOWANCE, minimum value is 2Ø.
*2 For EDGE ANGLE, minimum value is 2Ø.
*3 Dimension parameters are indicated in the table on the next page.

Dimension Parameter	Other than Precision Lathe Spec.				Precision Lathe Spec.			
	Metric System		Inch System		Metric System		Inch System	
	Minimum Value	Maximum Value	Minimum Value	Maximum Value	Minimum Value	Maximum Value	Minimum Value	Maximum Value
OD CLEARANCE	Ø	3Ø.ØØØ	Ø	1.1811	Ø	3Ø.ØØØØ	Ø	1.1811Ø
ID CLEARANCE	Ø	3Ø.ØØØ	Ø	1.1811	Ø	3Ø.ØØØØ	Ø	1.1811Ø
FACE CLEARANCE	Ø	3Ø.ØØØ	Ø	1.1811	Ø	3Ø.ØØØØ	Ø	1.1811Ø
BLANK INSIDE LENGTH IL	Ø	+9999.999	Ø	+393.7ØØ7	Ø	+9999.9999	Ø	+393.7ØØ78
DIMENSION ZERO SHIFT	-9999.999	+9999.999	-393.7ØØ7	+393.7ØØ7	-9999.9999	+9999.9999	-393.7ØØ78	+393.7ØØ78
PROGRAM ZERO SHIFT	-9999.999	+9999.999	-393.7ØØ7	+393.7ØØ7	-9999.9999	+9999.9999	-393.7ØØ78	+393.7ØØ78
NOSE-R CANCEL TRAVEL	Ø	1.ØØØ	Ø	Ø.Ø394	Ø	1.ØØØØ	Ø	Ø.Ø3937

12-3. MATERIAL DATA

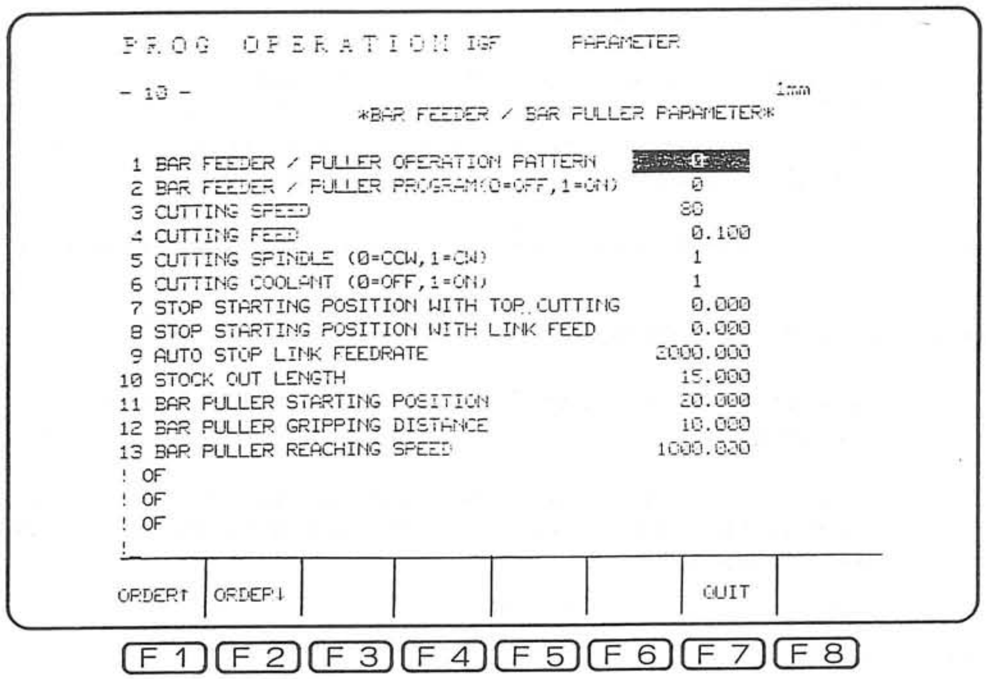
When designating data for the materials data display screen, units and upper and lower value limits apply in the same way as for the non-precision lathe specifications described in Section 4-2, "Material Data", in the Basic Manual (Publication No. 245Ø-E). When presetting this data through the graphic-edit function, minimum command unit data (for feedrate, infeed and finishing allowance) which have been designated differently in the graphic-edit function will be aligned to those settings for presetting. The minimum command units (Ø.1 μm, Ø.1 μm/rev, or Ø.ØØØØ1 in. and Ø.ØØØØ1 inch/rev) at this time are normally "Ø", but this digit may be commanded at will when engaged in graphic editing.

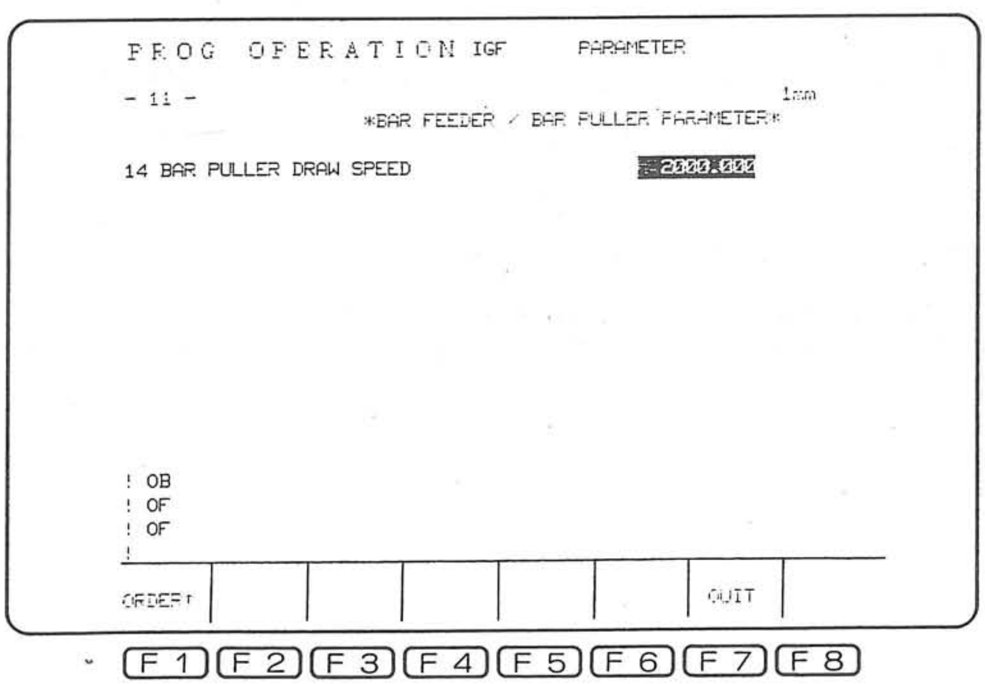
SECTION 13 BAR FEEDER AND BAR PULLER FUNCTION

When the machine is equipped with the bar feeder or bar puller, control of such automatic material handling devices is possible by setting proper parameters.

13-1. IGF PARAMETERS

The following IGF parameter pages are provided for the control of the bar feeder/puller. Input the data as required.





Description of parameters:

1 BAR FEEDER / PULLER OPERATION PATTERN (Ø - 4)

Ø ..... The machine is not equipped with bar feeder or bar puller.

1 ..... The machine equipped with bar feeder; both advance and retraction of bar feeder are controlled by M93
(models: PF-V4Ø, PF-V3, ANF65R, IBF-18NNC-2.Ø)

2 ..... The machine is equipped with bar feeder; advance and retraction of bar feeder are controlled by M93 and M92, respectively.
(models: MF-1, MF-3)

3 ..... The machine is equipped with bar feeder; chuck open command (M84) and chuck close command (M83) also function as the bar feeder advance and retraction command, respectively.
(models: IBF32-2H(3H), THB32, Sameca Sa)

4 ..... The machine is equipped with bar puller (spring collet type).

2 BAR FEEDER / PULLER PROGRAM (Ø, 1)

Ø ..... Control codes for bar feeder and bar puller are not output to the program.

1 ..... Control codes for bar feeder and bar puller are output to the program (the input screen is displayed at the first page for the IGF editing).

3 CUTTING SPEED

4 CUTTING FEED

5 CUTTING SPINDLE (Ø=CCW, 1=CW)

6 CUTTING COOLANT (Ø=OFF, 1=ON)

Set the data as required for these parameters.

7 STOP STARTING POSITION WITH TOP CUTTING

For the machine equipped with sizing stopper automatic feed function, input the sizing stopper positioning position for top cut operation.

8 STOP STARTING POSITION WITH LINK FEED

For the machine equipped with sizing stopper automatic feed function, input the sizing stopper positioning position for starting cutting.

9 AUTO STOP LINK FEEDRATE (mm/min)

Input the feedrate at which the sizing stopper is fed out in synchronization with the blank.

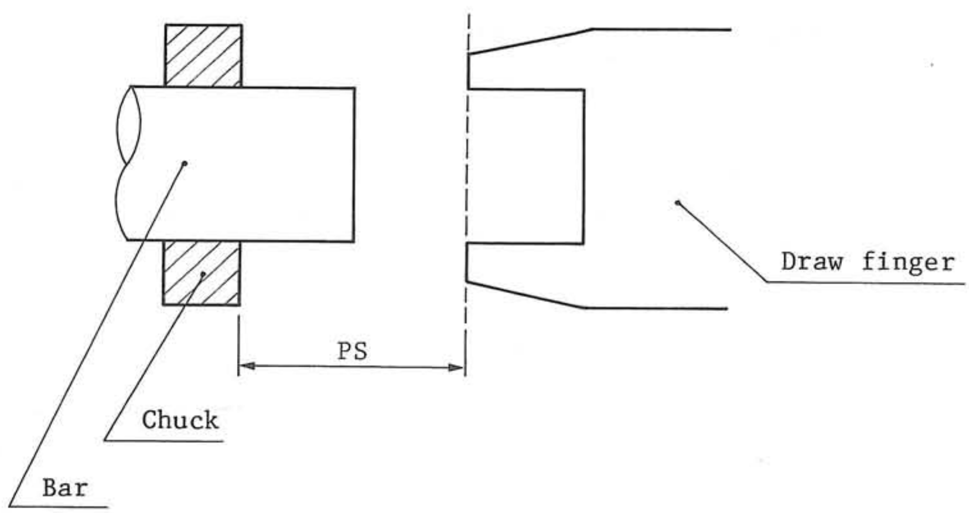
1Ø STOCK OUT LENGTH (FL)

Input the initial projection length of bar when bar material is changed.

11 BAR PULLER STARTING POSITION (PS)

Input the feed starting position of the draw finger.

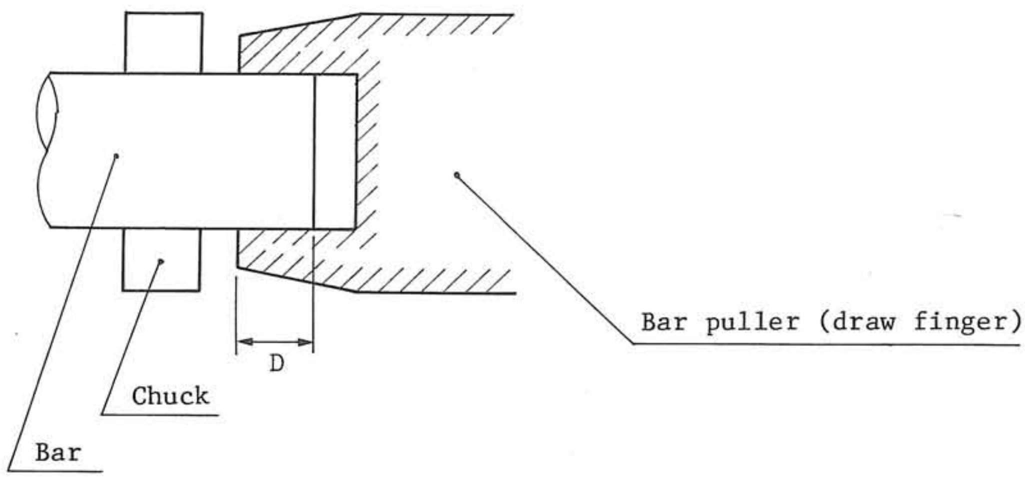
See the illustration below.



12 BAR PULLER GRIPPING DISTANCE (D)

Input the gripping length of material to be gripped by the draw finger.

See the illustration below.



13 BAR PULLER REACHING SPEED (mm/min)

Input the feedrate of the draw finger in which it is advanced to grip the bar.

14 BAR PULLER DRAW SPEED (mm/min)

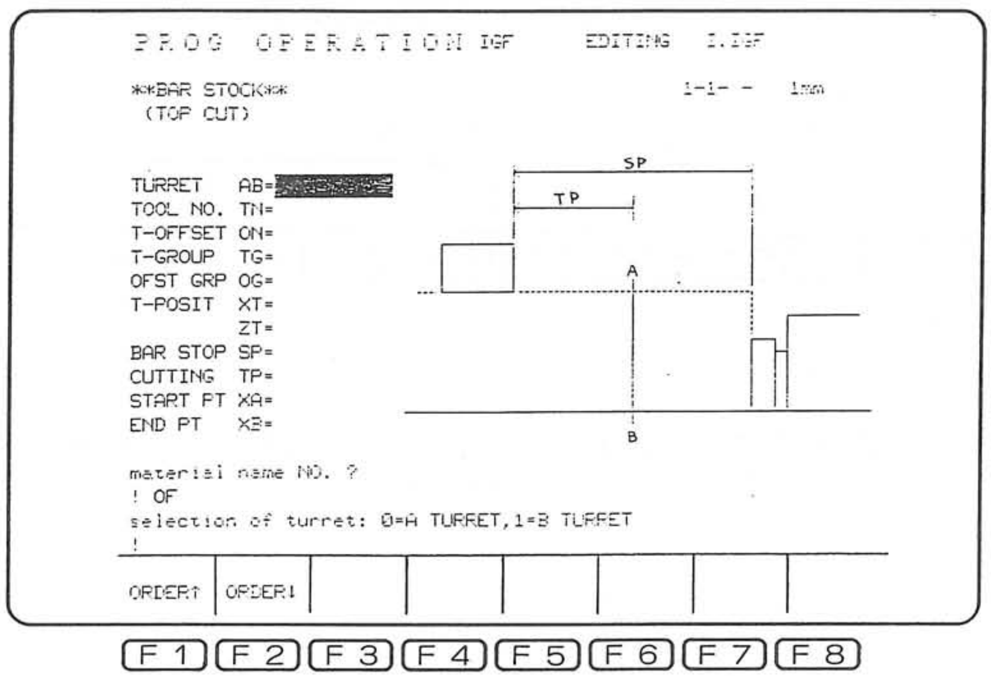
Input the feedrate of the draw finger in which it pulls out the bar.

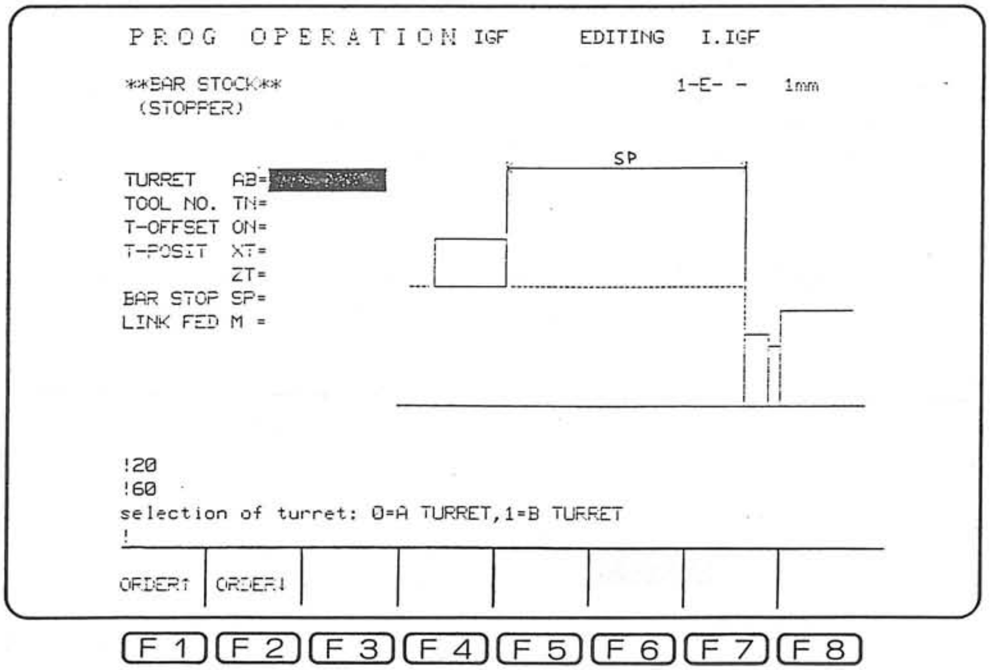
13-2. GRAPHIC EDITING FOR BAR FEEDER/PULLER SPECIFICATIONS

When "1" through "4" is set at No.1 of BAR FEEDER/PULLER PARAMETER screen, the system provides the screen where the input of the bar material is displayed when starting the graphic edit operation. Input of the data for the parameters at such screens makes it possible to control the bar feeder/puller by proper M code commands.

(1) Input of "1":

When "1" is input at No. 1, the displays as shown below are displayed for the input of data for top cutting and sizing stopper.



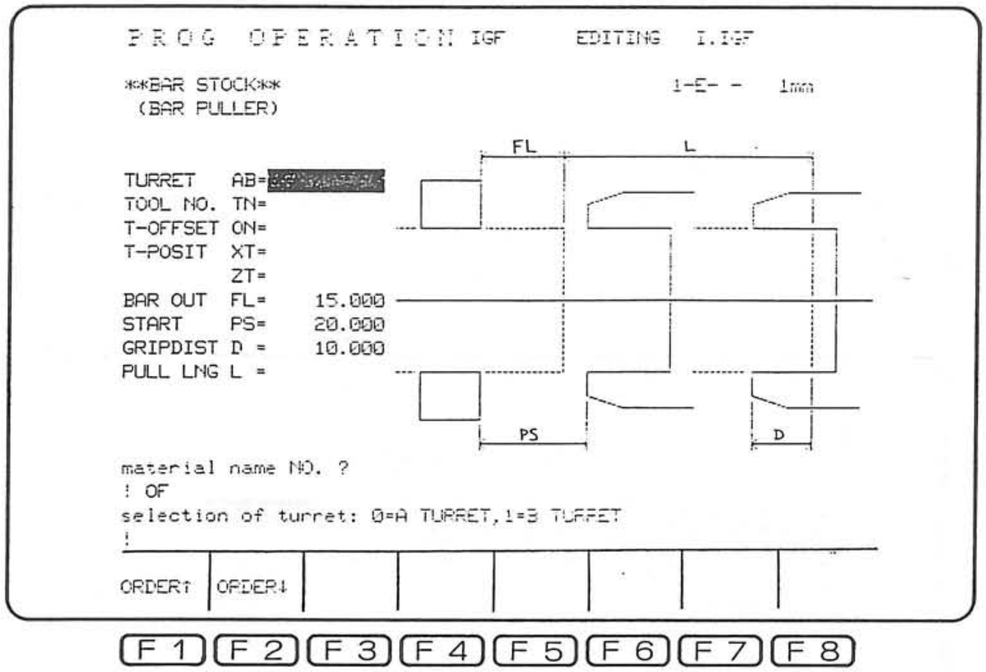


(2) Input of "2" or "3":

When "2" or "3" is input at No. 1, the display for the data input for the sizing stopper is displayed.

(3) Input of "4":

When "4" is input at No. 1, the display for the data input for bar puller is displayed as shown below.



* At the parameters BAR OUT, START and GRIPDIST in this page, the data set for the IGF parameters are displayed as the initial value.

Note 1: Parting off should be set as a single process.

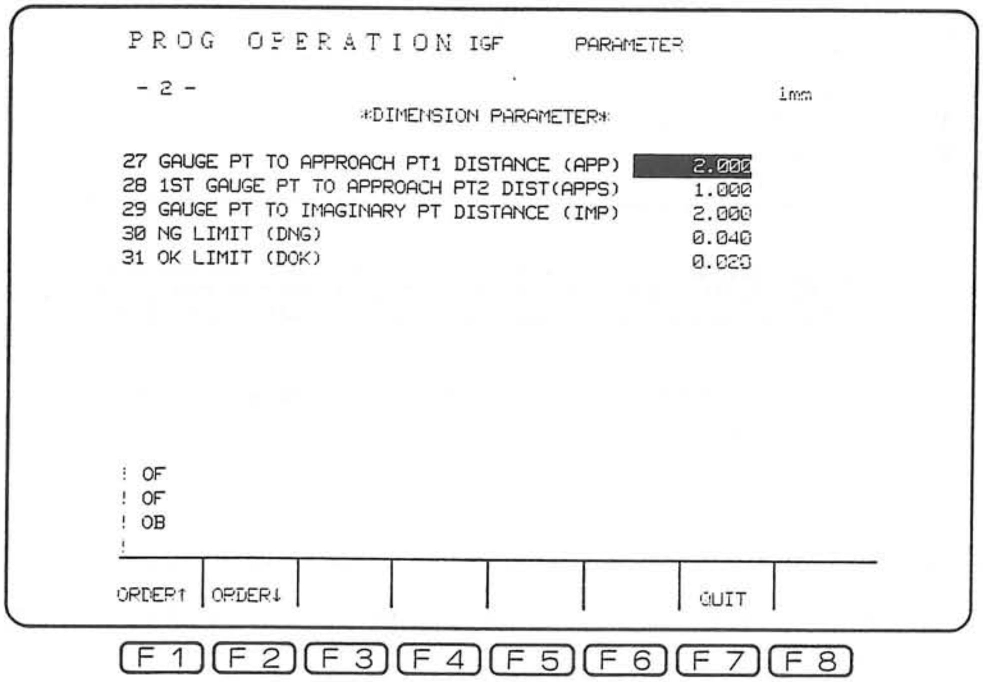
Note 2: The programs related with bar feeder or bar puller operations are not checked by the process test.

SECTION 14 IGF ON IN-PROCESS GAUGING FUNCTION SPECIFICATION

When the in-process gauging function is selected, gauging cycle programs can be easily made by designating measuring points as the measuring process in the graphic edit operation.

14-1. PARAMETERS USED FOR GAUGING CYCLES

The following parameter setting page is provided to set IGF dimension parameters related to the gauging function. Data is to be used as arguments when a gauging cycle subprogram is called from a machining program. Change the data as needed.



(1) Abbreviated codes indicated in ( ) at the end of each message are the arguments for which the data is set. For the details of these arguments, refer to the Instruction Manual for Automatic Work Gauging and Automatic Tool Gauging (Publication No. 27Ø9-E-R1).

(2) Parameters DNG and DOK can be set during the graphic editing operation regardless of the data set at the parameter setting screen.

14-2. MAKING GAUGING PROCESSES

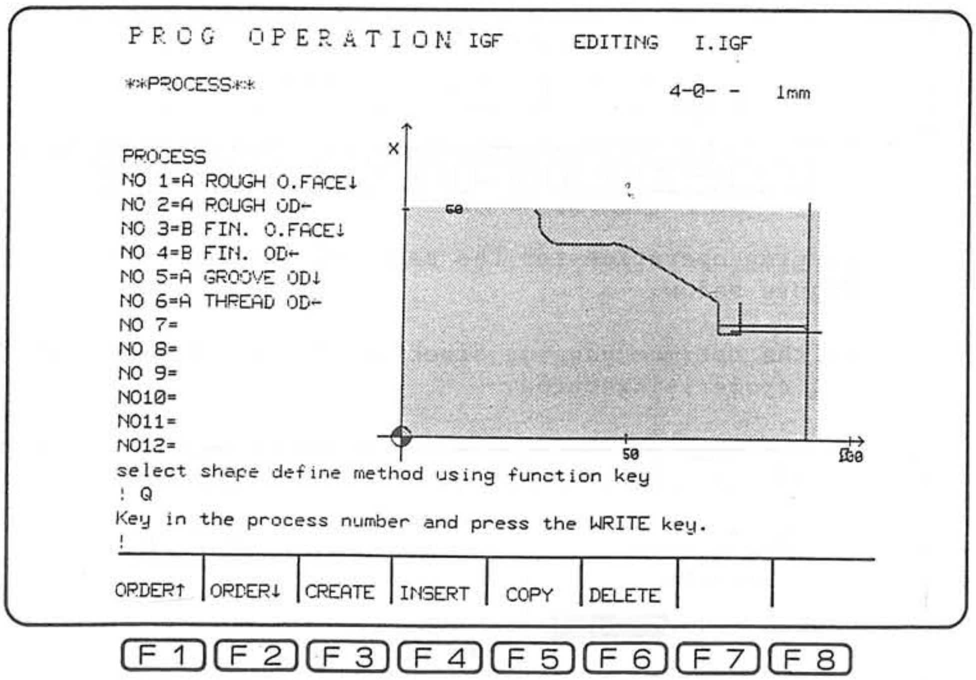
The gauging cycle is programmed taking one gauging operation as one process. To conduct the gauging cycle at different points, make a gauging process at each of these points.

14-2-1. Gauging Cycles for Rough, Finish, and Copy Processes

The procedure for making a gauging process are explained below. The process for which a gauging cycle is made with "FINISH OD←" of the shape explained in the Basic Section.

(1) After data input for the machining process is complete, input the data for the gauging cycle.

Press the function key [F3] (CREATE) and the WRITE key in the same manner as for the machining process.

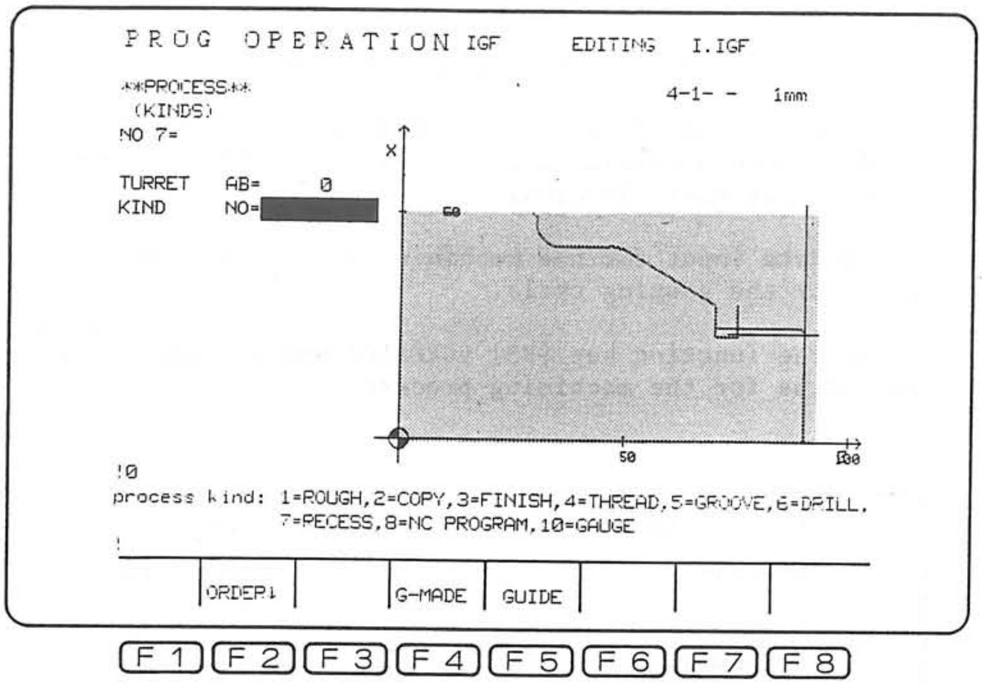


A gauging process may be created at any process and it may not necessarily be created only for the final process. It can be created after the completion of machining data input of a process for which the gauging cycle is to be executed.

The explanation below is given to make a gauging process for process "NO 4=B FIN. OD←".

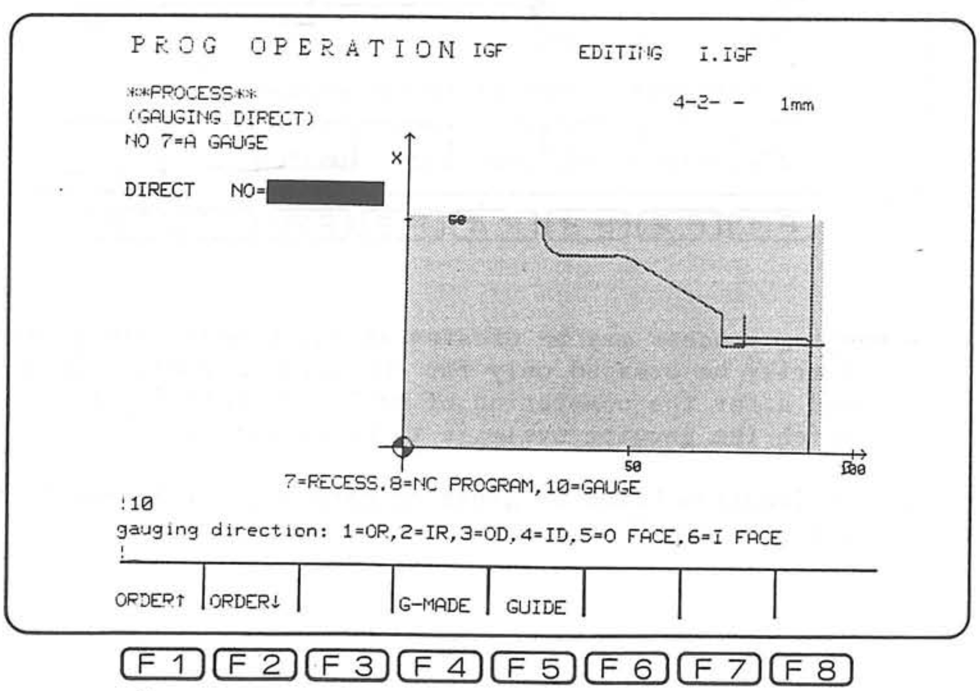
(2) With the two-saddle model, select the turret on which the sensor is installed.

Select "1Ø=GAUGE" for the process type.



(3) The gauging operation for the selected gauging direction is shown in the figure below.

Select the optimum gauging direction for the process for which the gauging cycle is executed.



<Gauging Operation Pattern>

Ⓣ : Turret indexing position

⊗ : Reference point

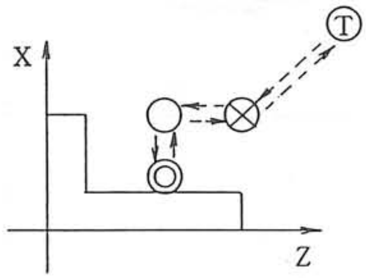
◎ : Gauging point

○ : Intermediate point, gauging cycle start point

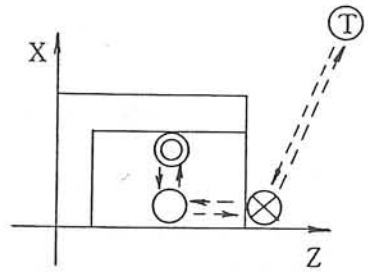
←—— : Axis movements in gauging cycle

←– – : Axis movements generated by IGF

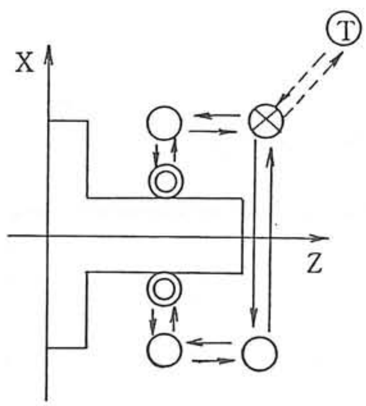
a) OR



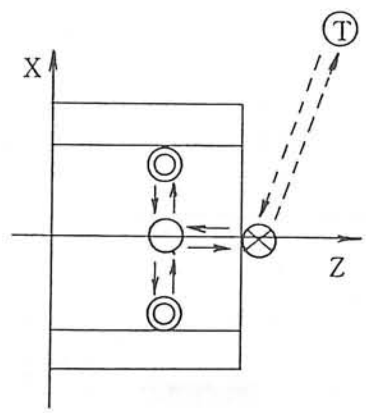
b) IR



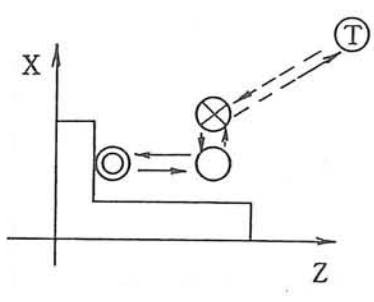
c) OD



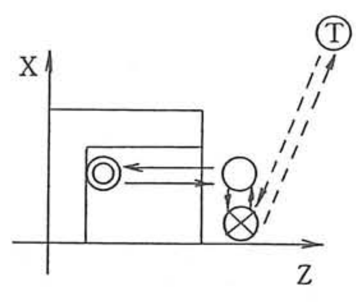
d) ID



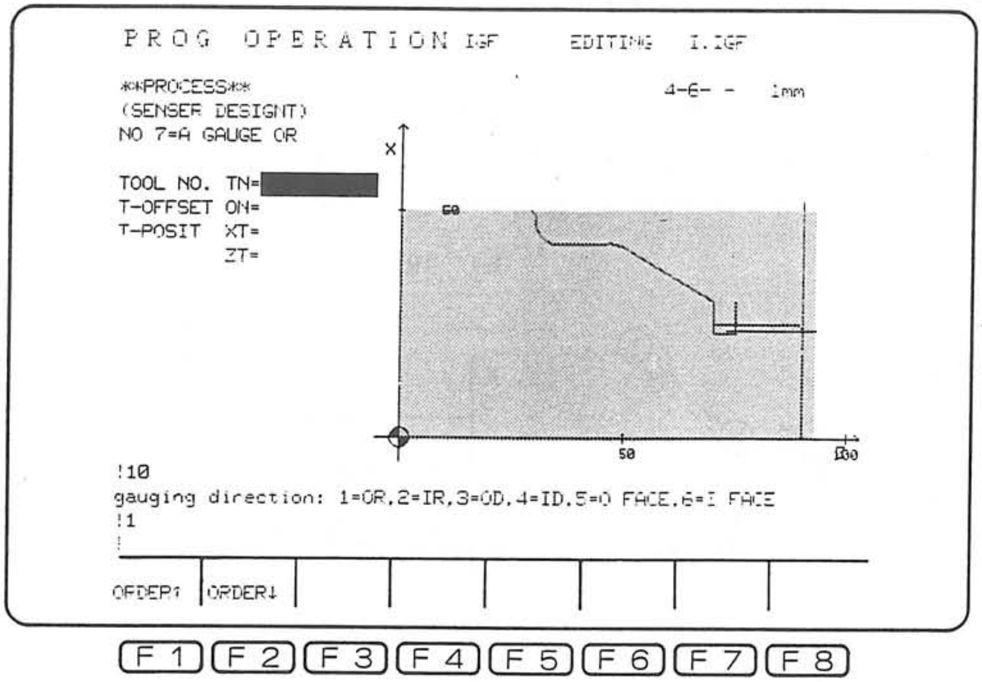
e) O FACE



f) I FACE

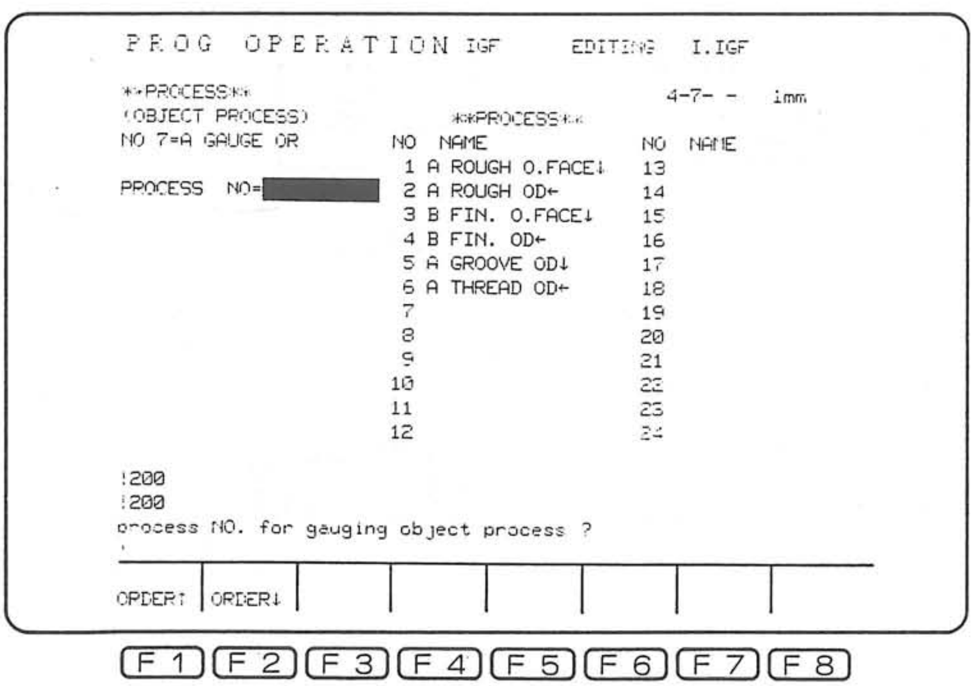


(4) Input the turret position number on which the sensor is installed, tool offset number of the sensor, and the turret index position.



(5) Select the process for which the gauging cycle is executed.

Processes selectable are only ROUGH, FINISH, COPY, and GROOVE. If any other process is selected, an error occurs.



In case the gauging direction set in step (3) and the process selected do not match each other*, an error occurs.

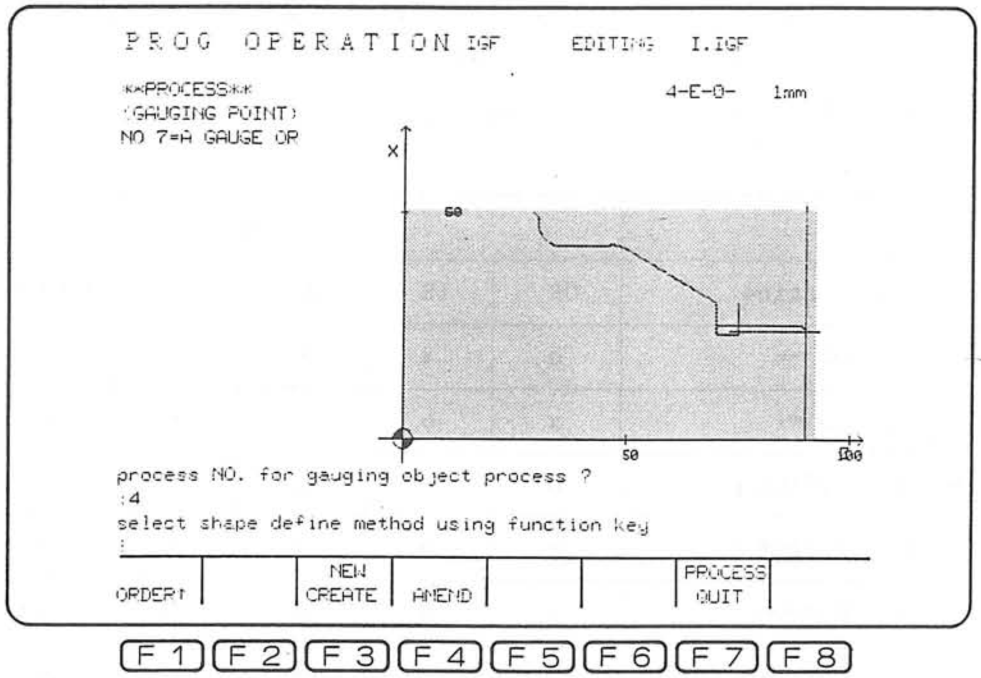
* The surface on which the gauging cycle can be made is not present in the selected gauging direction.

The allowable combinations of processes and gauging directions are shown in the table below.

Process		GAUGE					
Kind	Direction	OR	IR	OD	ID	O FACE	I FACE
1. ROUGH	1. OD ←	o	x	o	x	o	x
2. COPY	2. ID ←	x	o	x	o	x	o
3. FINISH	3. O.FACE ↓	o	x	o	x	o	x
	6. I.FACE ↑	x	o	x	o	x	o
	7. M.OD ←	o	x	o	x	o	x
	8. M.ID ←	x	o	x	o	x	o
	9. M.FACE ↓	o	x	o	x	o	o
	1Ø. M.OD →	o	x	o	x	o	x
	11. M.ID →	x	o	x	o	x	o
	12. M.FACE ↑	x	o	x	o	o	o
4. GROOVE	1. OD ↓	o	x	o	x	o	x
	2. ID ↑	x	o	x	o	x	o
	3. FACE ←	o	x	o	x	o	o
	4. WIDE OD ↓	o	x	o	x	o	x
	5. WIDE ID ↑	x	o	x	o	x	o
	6. WIDE FACE ←	o	o	o	x	o	o
	7. V OD ↓	o	x	o	x	x	x
	8. V ID ↑	x	o	x	o	x	x
	9. V FACE ←	x	x	x	x	o	o

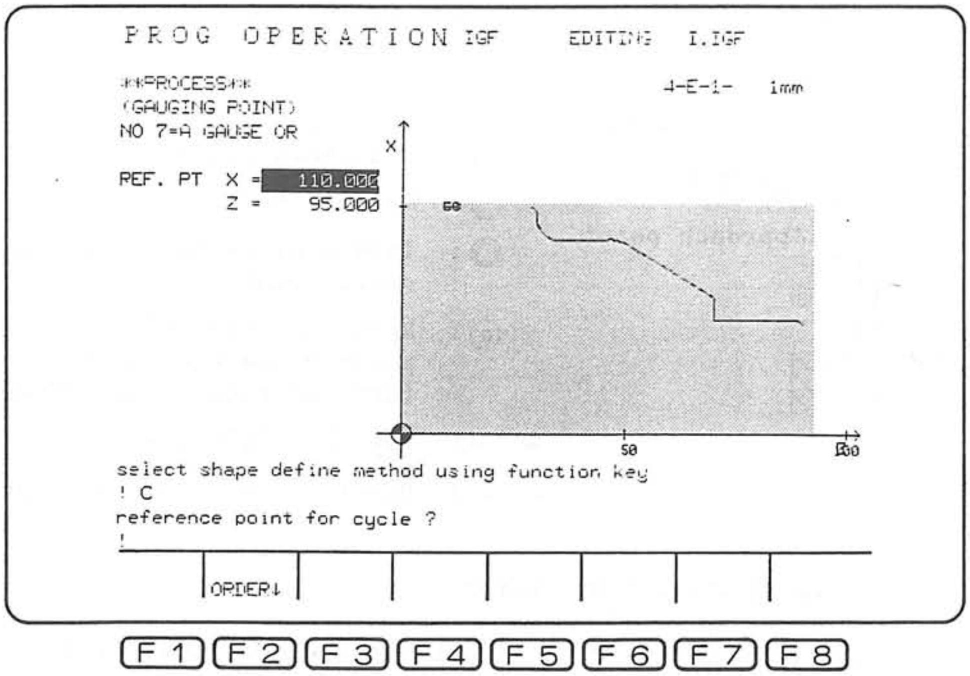
o: Gauging possible
x: Gauging impossible

(6) Press the function key [F3] (NEW CREATE) to set the gauging points.



(7) Set the reference point.

The reference point corresponds to the position indicated by ⊗ in the gauging operation chart given in step (3) above.



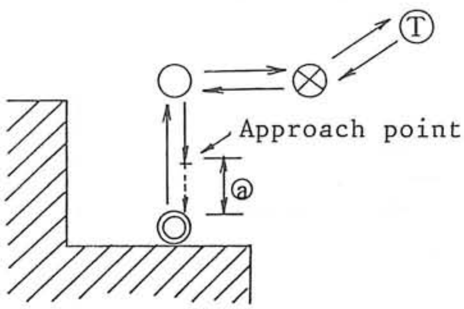
Default of the reference point coordinates:

X-axis
- For inside diameter gauging .... Ø
- For outside diameter gauging ... BLANK OUTSIDE DIA OD + OD CLEARANCE

Z-axis .... BLANK FACE + FACE CLEARANCE

Cautions:

In the gauging process setting screen, only the reference point and gauging points are set. The gauging cycle start point is automatically determined in the following manner according to the set reference point and gauging point.



Ⓣ : Turret indexing position

⊗ : Reference point

○ : Gauging point

◎ : Intermediate point, gauging cycle start point

ⓐ : Relative value of the gauging target point value and the approach point (APP: parameter set value)

←—— : Rapid positioning

←---- : Positioning in cutting feedrate

Gauging cycle start point:

a) When the gauging direction is O (outside) or I (inside):

Start point (X, Z)
= (X coordinate of reference point, Z coordinate of gauging point)

For OD:

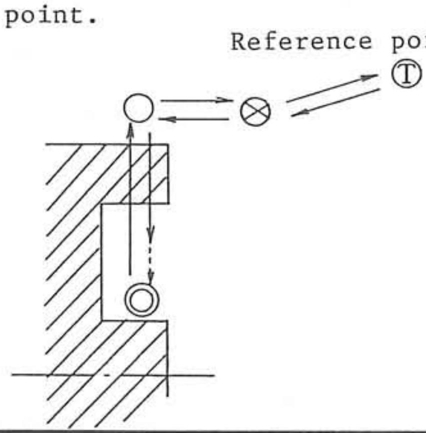
Start point (X, Z)
= (Ø of program coordinate, Z coordinate of gauging point)

b) When the gauging direction is FACE:

Start point (X, Z)
= (X coordinate of reference point, Z coordinate of gauging point)

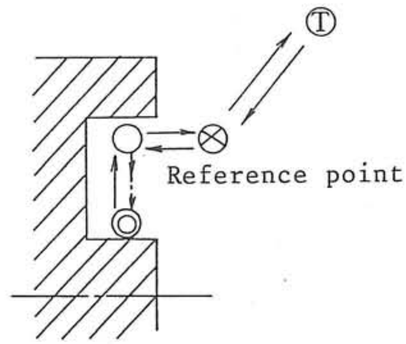
Therefore, depending on the workpiece shape, interference may occur if the gauging cycle is executed using the preset reference point.

Example:



If the selected reference point is not correct, interference occurs during sensor movement from the start point to the gauging point. Therefore, the reference point must be reset at a position where axis movement from the reference point to the gauging point will not cause interference.

Example: To avoid interference, set the reference point at a position as illustrated at left.

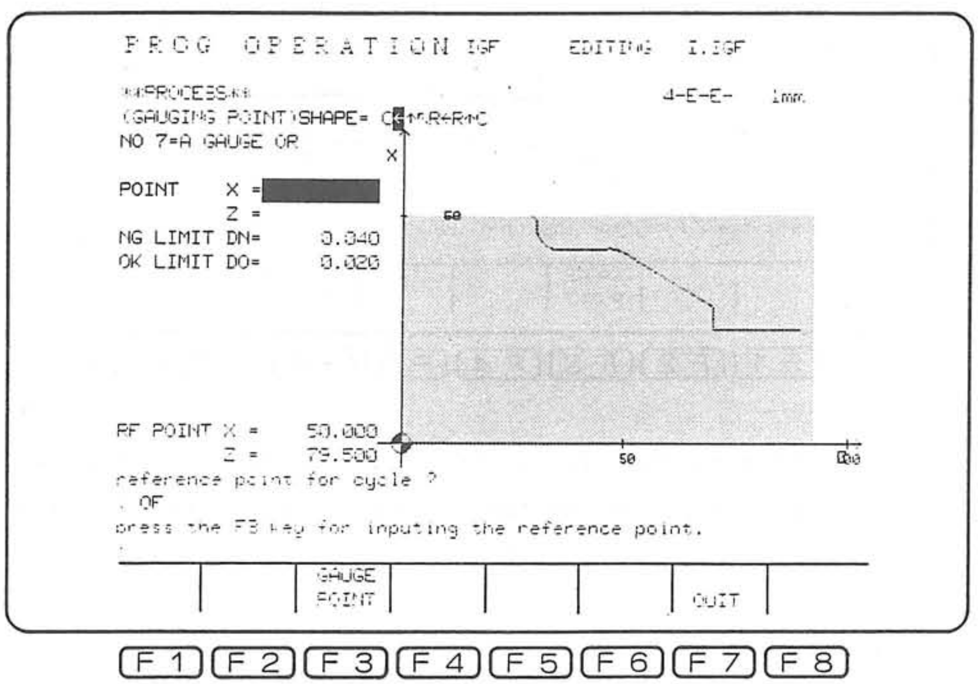


(8) Set the gauging point.

The gauging point can be set by the following two methods.

a) Automatic setting using the reference point
b) Direct setting by keying in the coordinate values

Select the optimum method depending on the point at which gauging is to be executed.



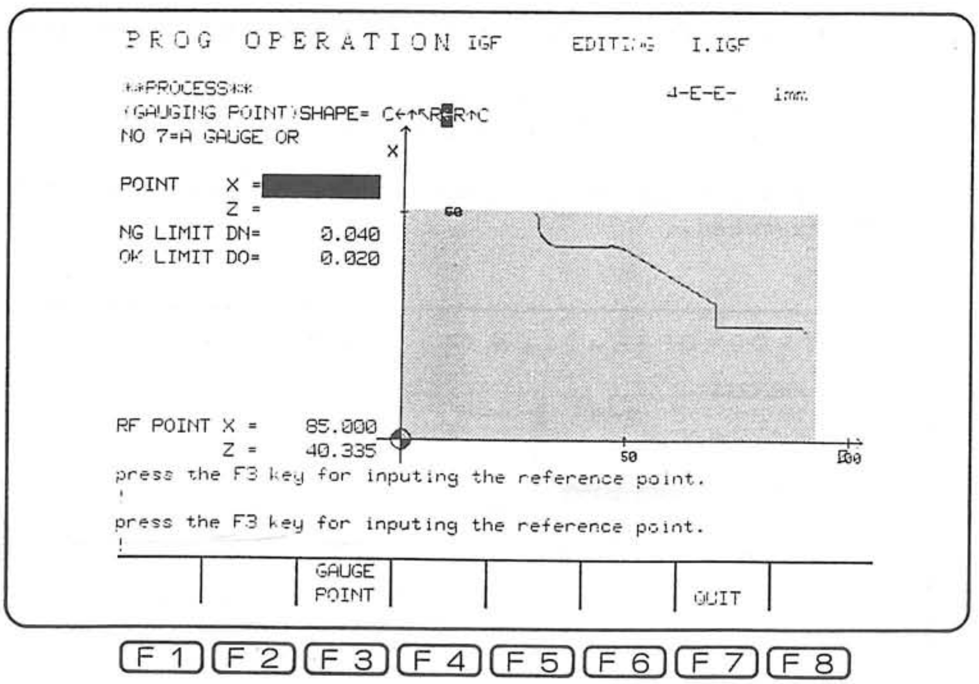
a) Automatic setting using the reference point

A flickering point will appear on the shape displayed on the screen. This point appears at the mid point between the start and end points of the shape element intersecting at right angles to the designated gauging direction.

In the example, the shape elements used to define the shape are "C ← ↑ ↖ ← R ↑ C". Because "OR ↓" is selected as the gauging direction, a flickering point appear at the mid point between the start and end points of the shape element drawn by "←". Flickering points do not appear on other shape elements.

In this example, two elements represented by "←" are displayed. When the shape display screen is displayed first, the cursor is located on the left side "←" and a flickering point appears on the corresponding shape element.

Pressing the cursor control key [→] shifts the cursor to the right side "←", and the flickering point appears on the corresponding shape element.



Similarly, the flickering point can be displayed on the required shape element using the cursor control keys [→] and [←].

The coordinate values of such a point are displayed in the lower left area of the screen. This is the reference point. To use the reference point as the gauging point, proceed as follows:

1) Move the cursor so that the flickering point appears at the required shape element.

2) Shift the cursor to POINT X or Z.

3) Press the function key [F3] (GAUGE POINT).

The coordinate value displayed as RF POINT automatically set at the coordinate value of POINT. Conduct steps 1), 2) and 3) for both X and Z.

When the gauging point has been set, an "x" appears at the flickering point position. This point is taken as the gauging point.

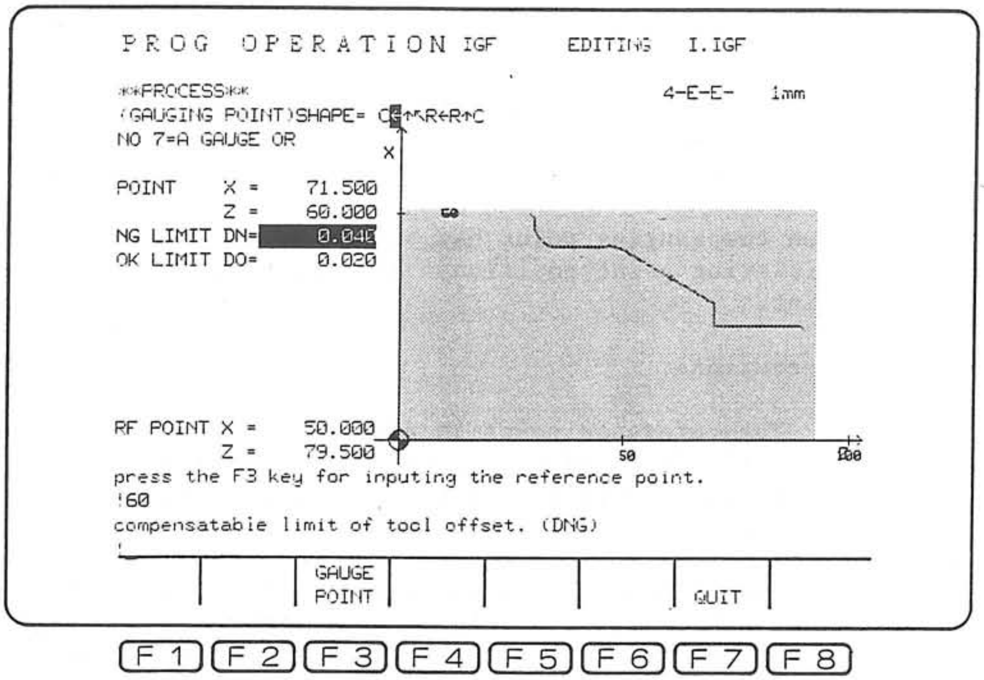
Precautions:

1) The reference point is displayed only on the shape element which is at right angles to the designated gauging direction. The reference point is not displayed on an arcs or tapers.

2) When there is no shape element on which a reference point can be displayed, the message "there is no gauging reference point" appears in the lower left corner of the screen. In this case, the reference point and reference point data are not displayed.

b) Direct setting by keying in the coordinate values

To conduct the gauging cycle at a point other than the reference point, directly input the required gauging point coordinate values from the keyboard. After the coordinate values are input, an "x" appears at the corresponding position.



(9) OK and NG limit setting

The NG limit establishes the allowable error from the target value (commanded value), and if the difference between the target value and the actual measured value exceeds this value, an alarm occurs.

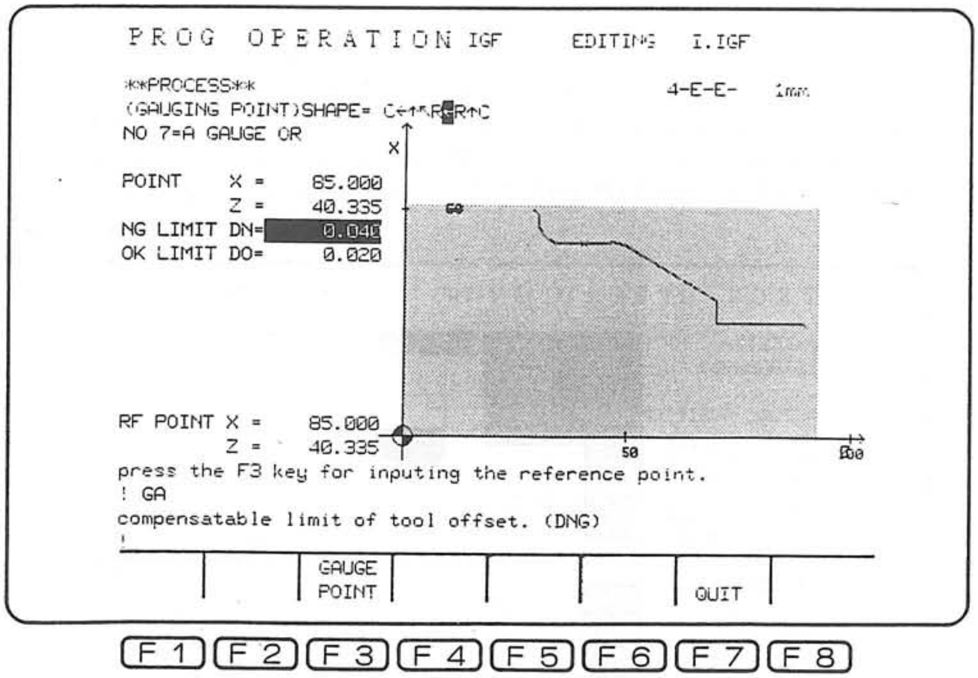
The OK limit establishes the compensation activation limit. That is, if the error between the target value and the actual measured value exceeds this value, compensation is made.

For the limit setting values, those preset at the parameter setting screen are displayed as defaults. If a value change is required, directly key in the value from the keyboard.

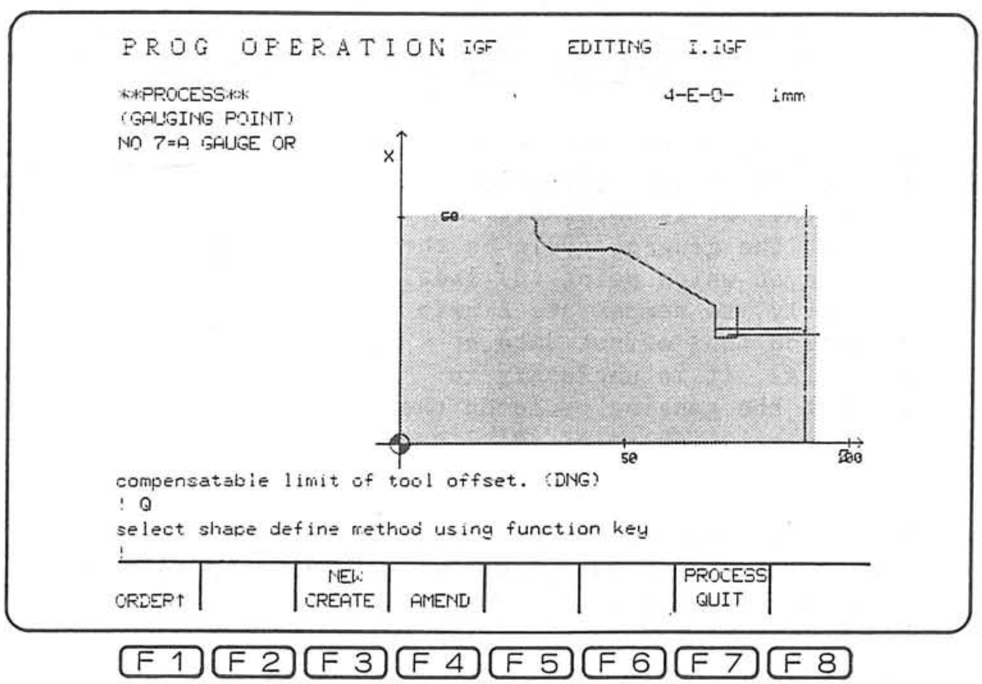
For details of OK and NG limits, refer to the Instruction Manual for Automatic Work Gauging and Automatic Tool Gauging (Publication No. 27Ø9-E-R1).

(1Ø) Press the function key [F7] (QUIT) to stop the gauging point setting mode.

The screen at right shows the display example when the reference point on the second "←" shape element is automatically taken as the reference point.



(11) Press the function key [F7] (QUIT) to stop the gauging process creation mode.

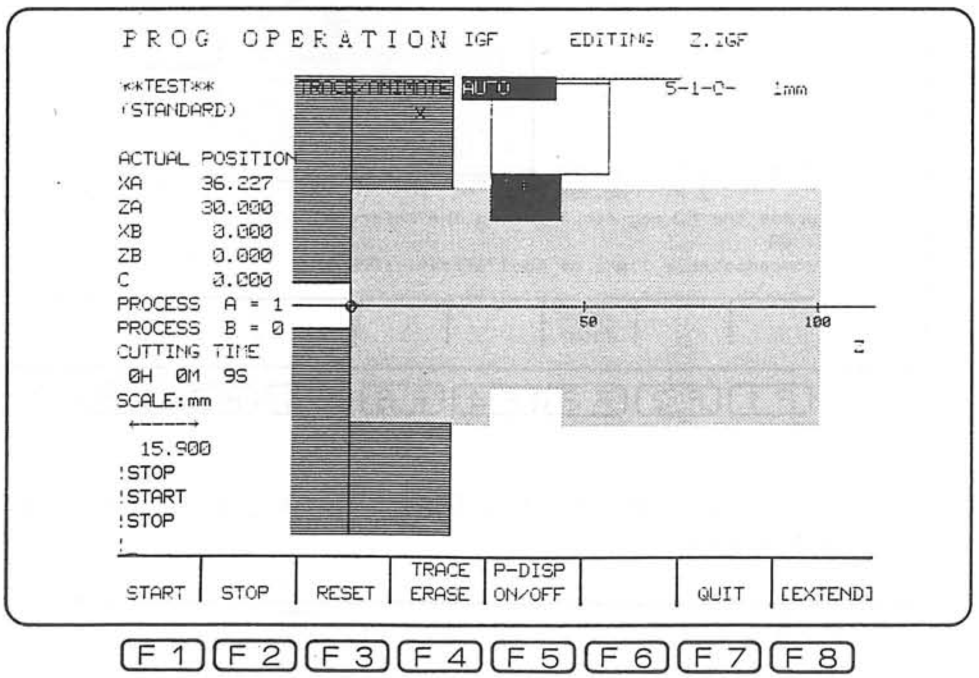


14-2-2. Gauging Cycle for Grooving Process

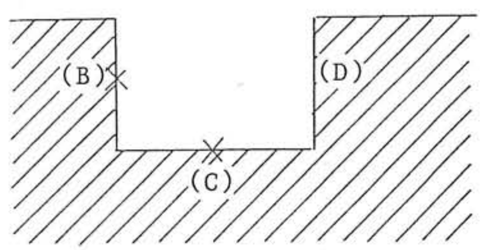
For the grooving process, operations required to create the gauging process are basically the same as used for the gauging process creation for Rough, Finish and Copy. The only difference is the display of the reference point for setting the gauging point explained below.

(1) Groove (single groove)

For a grooving tool, the tool offset data is set for point (A) with the corresponding tool path displayed. In this case, the groove width is identical to the grooving tool width. Therefore, the tool offset value calculated after the execution of the gauging cycle is the offset value for point (A) of the grooving tool.



To compensate the tool offset data of the X axis direction at point (A), it is necessary to conduct the gauging cycle on the surface on which point (C) lies. Similarly, to compensate Z axis direction tool offset data at point (A), it is necessary to conduct the gauging cycle on the surface on which point (B) lies.



Because the surface on which point (D) lies is determined by the tool width, it is not directly influenced by the tool offset data. Thus, the tool offset is not used for this surface.

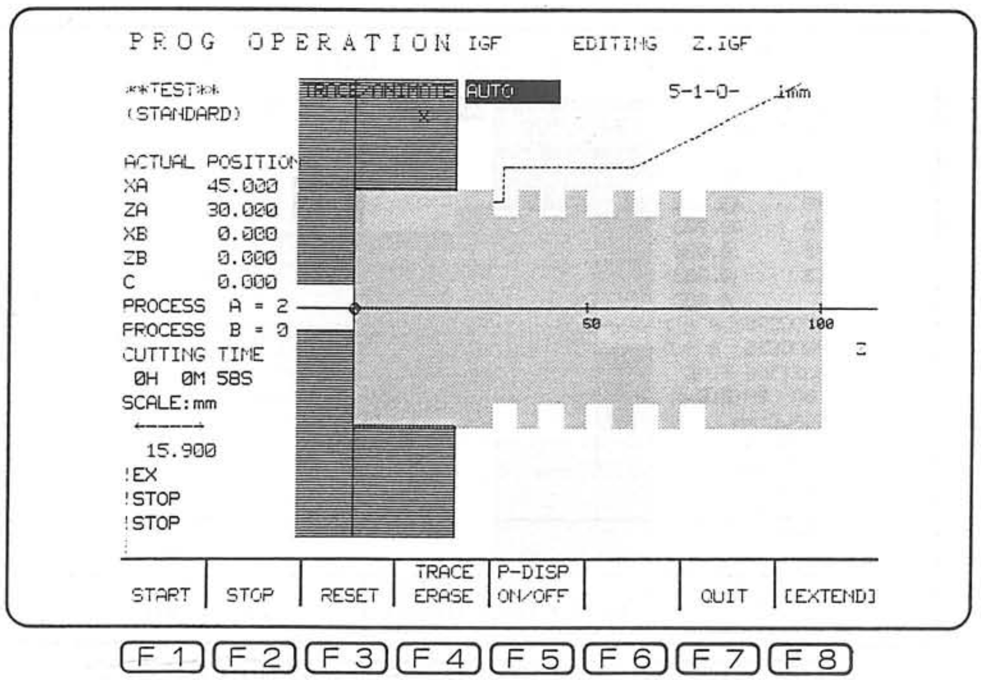
Display of a reference point for the setting of the gauging point is given on surface (C) when the gauging direction is "↓", and on surface (B) when the gauging direction is "←".

In the same manner, a reference point is displayed at the center of the surface which is directly influenced by the tool offset in end face grooves and ID grooves.

(2) Multiple grooves

For the multiple groove machining operation, the first groove position is designated. The other grooves are machined in reference to the first groove by designating the shift amount.

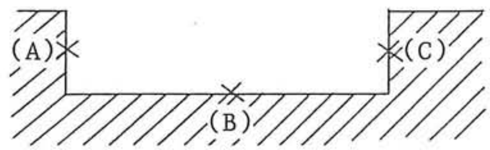
Therefore, the gauging cycle should be conducted for only one groove. In the IGF gauging process, a reference point is displayed only at the first groove.



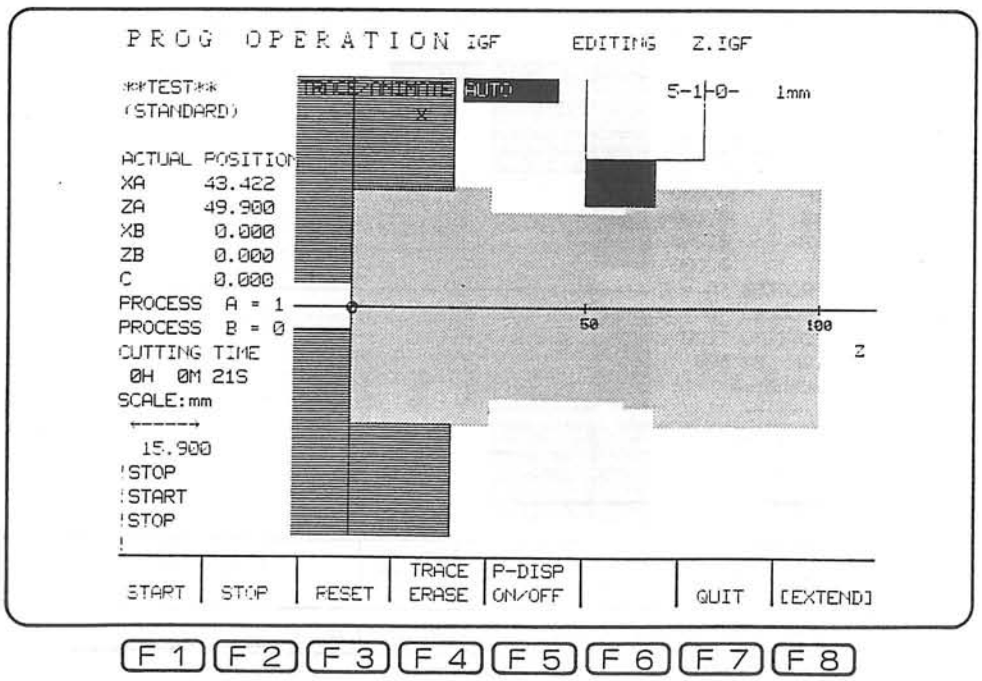
The reference point is displayed at the center of the surface which is directly influenced by the tool offset. Although the reference point is displayed at the first groove only, the gauging cycle can be conducted on other groove by directly inputting the gauging point coordinate values from the keyboard.

(3) Wide groove

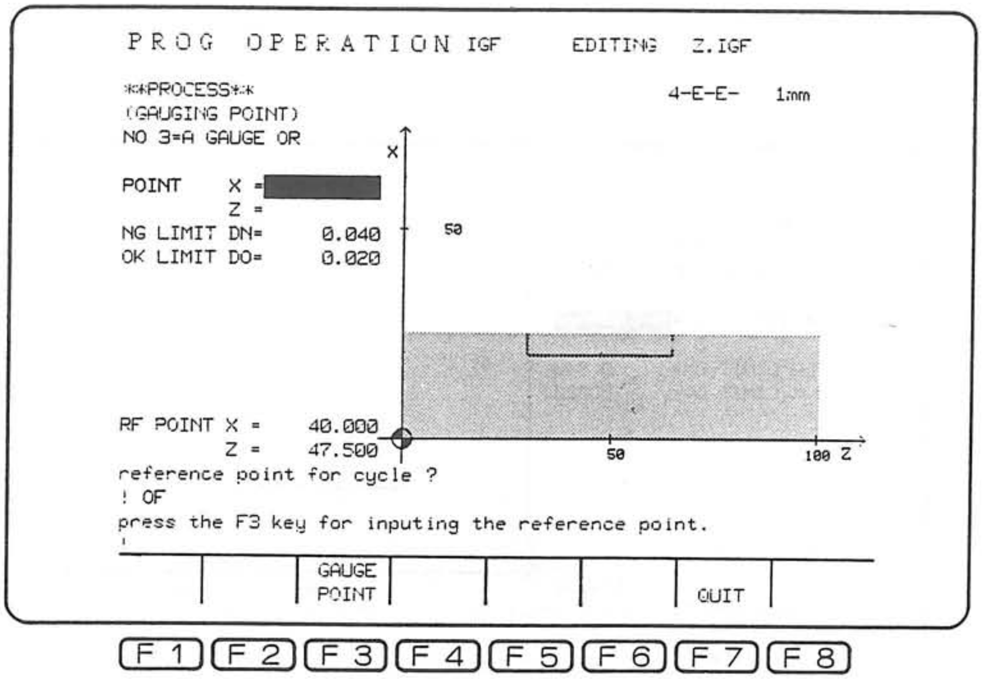
In wide groove machining, the tool offset data at point (D) (tool left edge) is used for machining surfaces (A) and (B) and the tool offset data at point (E) (tool right edge) is used for machining surface (C).



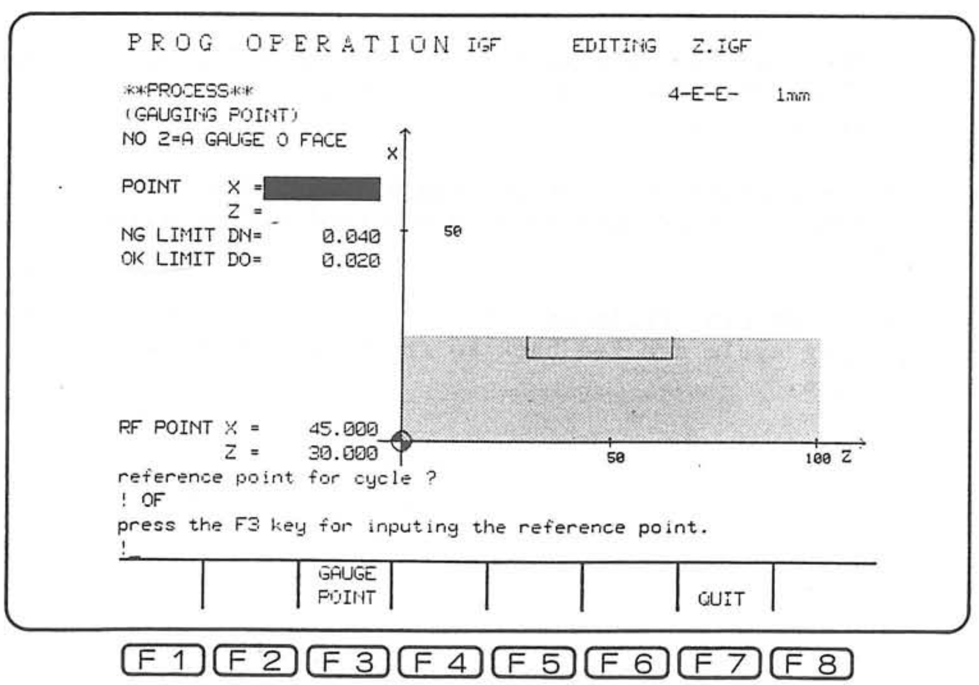
This means that reference points are displayed on all of these machined surfaces because all are directly influenced by the offset data.



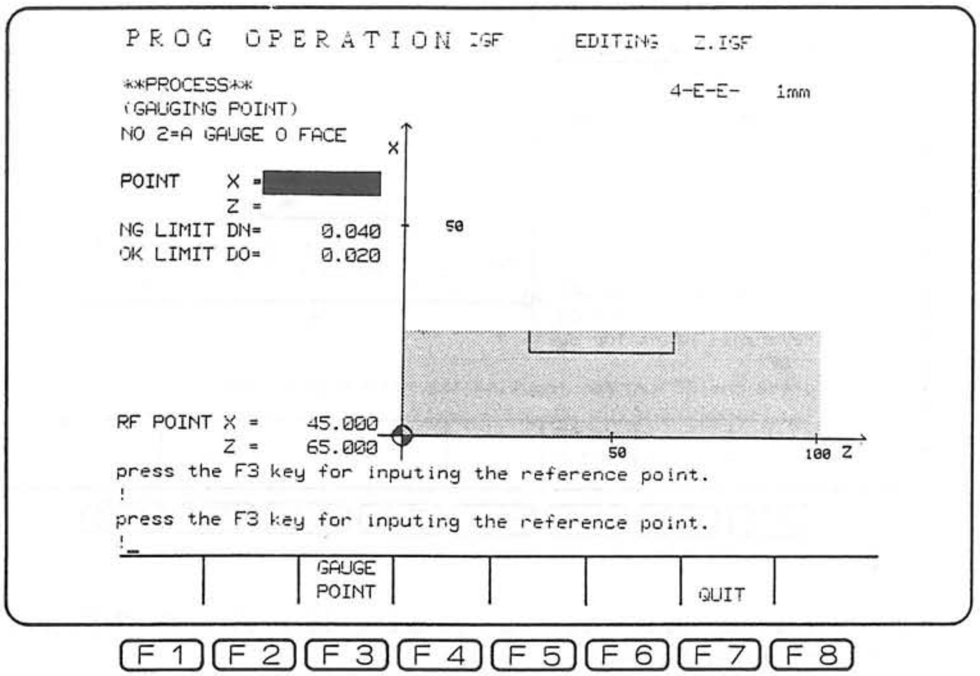
The reference point for the gauging direction "↓" is point (B') as shown below.



The reference point for the gauging direction "←" is point (A') or (C') as shown below.



The first reference point is always displayed at point (A'). To display reference point (C'), press the cursor control key [←] . To return the display to reference point (A'), press the cursor control key [←] .



In the gauging cycle at reference point (A'), the results of the gauging cycle are fed back to the tool offset value (Z) at tool point D.

In the gauging cycle at reference point (B'), the results of the gauging cycle are fed back to the tool offset value (X) at tool point D.

In the gauging cycle at reference point (C'), the results of the gauging cycle are fed back to the tool offset value (Z) at tool point E.

14-3. PROCESS EDIT OPERATION PRECAUTIONS

The gauging process is automatically deleted when the process for which the gauging process is defined is changed or deleted.

(1) After a machining process change

If any of the TURRET, KIND, or DIRECTION of the machining process are changed, the corresponding gauging process is deleted. When the display is returned to the PROCESS screen after the changed process complete, the message "Deleted the gauging process" is displayed in the lower left area of the screen. This deletes the gauging process from the screen.

(2) After deletion

When a machining process is deleted by pressing the function key [F6] (DELETE) and the corresponding process number, it automatically deletes the gauging process.

The message "Deleted the gauging process" appears in the lower left area of the screen and the gauging process display disappears from the process list.

14-4. GAUGING PROCESS TEST

In the IGF process test, the gauging process is processed in the following manner.

(1) On the IGF TEST screen, only tool paths are displayed for the gauging process graphic display.

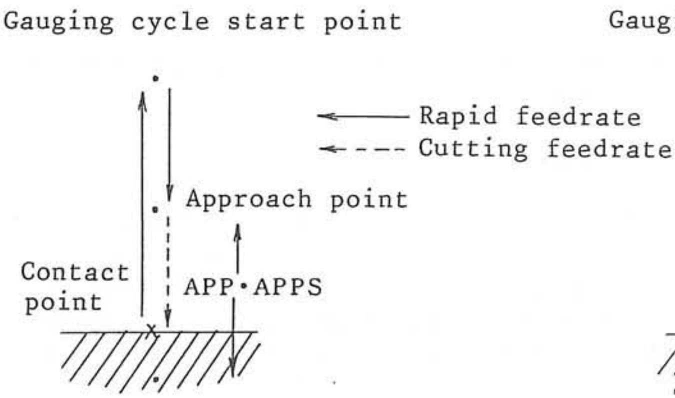
(2) The IF statement block created as the gauging cycle ON condition is not executed and ignored.

(3) Axis movement from the approach point to the gauging point is displayed as a rapid feedrate movement path. Actually, the gauging cycle is executed at a cutting feedrate within this distance.

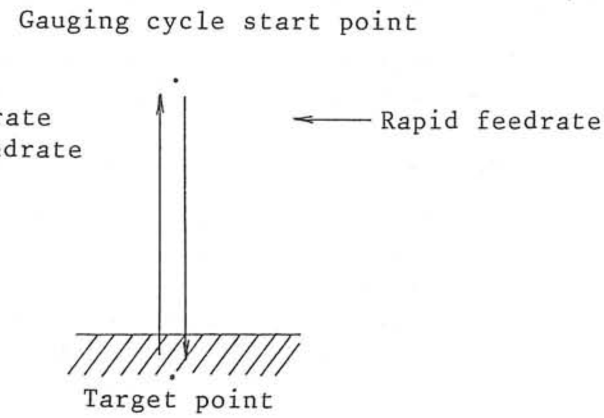
(4) Cutting time calculation for the gauging process is made in the following manner.

Gauging cycle time
= Positioning time
(from turret index position to gauging cycle start point)
+ Positioning time* (from gauging cycle start point to target point)

* Actual operations from the gauging cycle start point to the target point are as indicated in the figure at left. Calculation of the cycle time is made assuming the gauging cycle as shown at right.



Actual Operation



The Assumed Operation for Calculating the Cycle Time

Although an error is generated in axis movement from the approach point to the contact point, it is negligible in the sense that the distance between the approach point and the target point is small as set by the parameters (APP, APPS).

However, an error may not be negligible if such distance is re-set to a larger value.

14-5. NC GAUGING PROCESS PROGRAM

14-5-1. Configuration of the Gauging Process Program

The gauging process is converted into an NC program whose configuration is shown below.

```
                 Animation Data

NØØØ1            Machining Process
                 (Process for which
                  gauging process is
                  designated)

                 If the condition to execute the gauging cycle is not
                 satisfied, the process advances to the next process.

                 IF[VDIN[32]EQ Ø]NMSØ1
                 Sensor indexing
                 Positioning at gauging cycle start point              ---(1)
                 Call of gauging cycle subprogram
                 (OWXMA, ODOAA, etc.)
                 Retraction of turret to the turret index position

NMSØ1            Next Process

                 Call of gauging cycle ON condition cancel
                 subprogram (OMSSF)                                    ---(2)
                 Call of gauging cycle ON counter check
                 subprogram (OCNCK)

                 MØ2
```

Program areas in [double-lined box] are the parts converted by the creation of the gauging process.

(1) ..... Created according to gauging process

(2) ..... Always created at the end of a program even if the gauging process is designated only once.

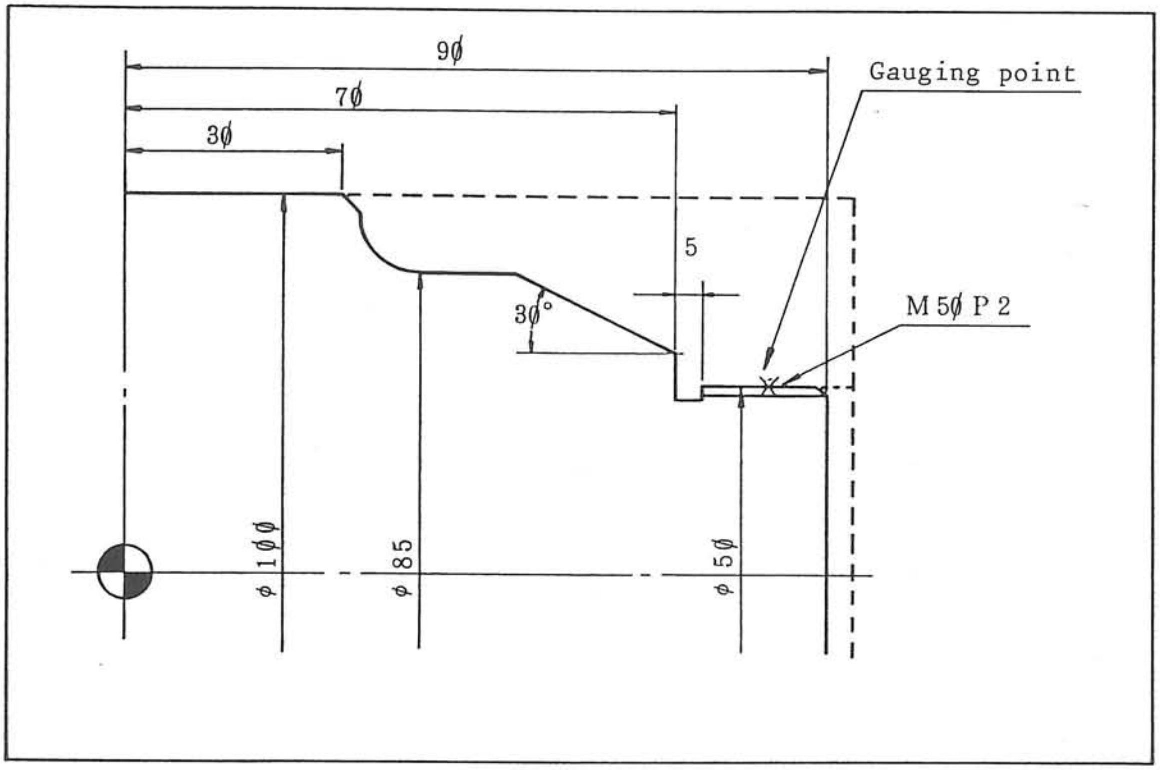
14-5-2. Example Program

```
NØØØ1 G13
NØØØ2 GØØ X    5ØØ       Z    8ØØ
NØØØ3 G5Ø                           S35ØØ
NØ1ØØ G97                           S 562 M41 MØ3 MØ8
NØ1Ø1 GØØ X    11Ø       Z     92   TØ3Ø3Ø3
NØ1Ø2     X    1Ø2
NØ1Ø3 G96 G11Ø                      S 18Ø
NØ1Ø4 G87 NØ1Ø5
NØ1Ø5 G81
NØ1Ø6 GØØ X     48
NØ1Ø7 GØ1                Z     9Ø   G42 F Ø.18
NØ1Ø8     X     5Ø       Z     89
NØ1Ø9                    Z     7Ø
NØ11Ø     X     6Ø
NØ111     X     82.321   Z     5Ø.67
NØ112 GØ3 X     85       Z     45.67  I -8.66
NØ113 GØ1                Z     35
NØ114 GØ2 X     95       Z     3Ø   I 5
NØ115 GØ1 X     98
NØ116     X    1ØØ       Z     29
NØ117 G4Ø                           K -Ø.Ø1
NØ118 G8Ø
NØ119 GØØ X    1Ø2
NØ12Ø                    Z     92
NØ121     X    11Ø
NØ122 G97                           S 562 MØ5 MØ9
NØ123     X    5ØØ       Z    8ØØ   TØ3ØØ
NØ2ØØ IF [VDIN[32] EQ Ø] NMSØ1           ←Judgment of gauging cycle ON conditions
NØ2Ø1 GØØ X    11Ø       Z     95   T121212←Sensor indexing
NØ2Ø2                    Z     79.5        Positioning at gauging cycle start point
NØ2Ø3 CALL OWMXA MSP=5Ø APP=2 APPS=1 IMP=2 DNG=Ø.Ø4 DOK=Ø.Ø2 TOFN=Ø3←
NØ2Ø4                    Z     95                  Call of gauging cycle subprogram
NØ2Ø5     X    5ØØ       Z    8ØØ   T12ØØ ←Retraction of turret to the turret
NMSØ1                                      index position
NØ3ØØ GØØ X    5ØØ       Z    8ØØ
NØ3Ø1 G97                           S 6Ø1 MØ3 MØ8
NØ3Ø2     X    11Ø       Z     7Ø   TØ9Ø9Ø9
NØ3Ø3     X     64
NØ3Ø4 G73 X     46       Z     7Ø   D 4     L 2Ø      F Ø.1    E Ø.2
NØ3Ø5 GØØ X    11Ø
NØ3Ø6                               MØ9
NØ3Ø7     X    5ØØ       Z    8ØØ   TØ9ØØ
NØ4ØØ G97                           S 672 MØ8
NØ4Ø1 GØØ X    11Ø       Z     93   TØ7Ø7Ø7
NØ4Ø2     X     6Ø
NØ4Ø3 G71 X     47.4     Z     73   H 2.6 D 1.Ø38 U Ø.Ø6  B 6Ø   F2   M22
&M73
NØ4Ø4 GØØ X    11Ø
NØ4Ø5                               MØ5 MØ9
NØ4Ø6     X    5ØØ       Z    8ØØ   TØ7ØØ
NØ4Ø7 G13
```

```
NØ4Ø8                          PØØ1Ø
NØ4Ø9 CALL OMSSF                       ← Call of gauging cycle ON condition
                                         cancel subprogram (A-turret side)
NØ41Ø CALL OCNCK                       ← Call of gauging cycle ON counter
                                         check subprogram
NØ411 G14
NØ412                          PØØ1Ø
NØ413 CALL OMSSF
NØ414 MØ2
```

Description of Example Program:

The example program consists of the FIN. OD←, GROOVE OD↓, and THREAD OD← processes.



Drawing of Machining Program Created in the Basic Section

NØ1ØØ - NØ123: FIN. OD←
NØ3ØØ - NØ3Ø7: GROOVE OD↓
NØ4ØØ - NØ4Ø6: THREAD OD←
NØ2ØØ - NMSØ1: Gauging process ((1) in 14-5-1)

This is the gauging process for the FIN. OD←. The gauging point is the point marked by an "x" in the diagram above. In NØ2ØØ, the gaugng cycle ON conditions are checked. If the conditions are not satisfied, the process advances to the next process "GROOVE OD↓".

The NØ4Ø7 - NØ413 blocks cancel the gauging cycle ON conditions as well as check the gauging cycle ON counter. These blocks are always created even if when a gauging process is designated only once. ((2) in 14-5-1).

Precautions:

(1) Data at VDIN[32] in the first block of (1) is automatically set to "1" when the gauging cycle ON condition counter counts up, or the GAUGING CYCLE ON button on the operation panel is pressed and the gauging cycle is executed.

(2) Condition check for the initial workpiece gauging and the initial workpiece gauging program for the tool life management specification are not created by the graphic edit function.

To execute the initial workpiece gauging, add the following information in the editing operation.

```
        :
        :                 Check if the automatic tool change has been conducted
                          due to tool life expiration
NØØØ1   CALL OTRCK
                                    Proceed to the machining process when the
        IF[VDIN[33]EQ Ø]NMS99       initial workpiece machining or initial
                                    workpiece gauging cycle ON conditions are
                                    not satisfied.
        Initial workpiece machining program
          (program containing larger stock)

        Indexing sensor

        Positioning at gauging cycle start point

        Calling the gauging cycle subprogram
          (To allow tool offset data compensation to be conducted
           by setting a large ±OK zone)

        Retraction to turret index position

NMS99   Machining process
            :
            :
        Gauging
            :
            :
```

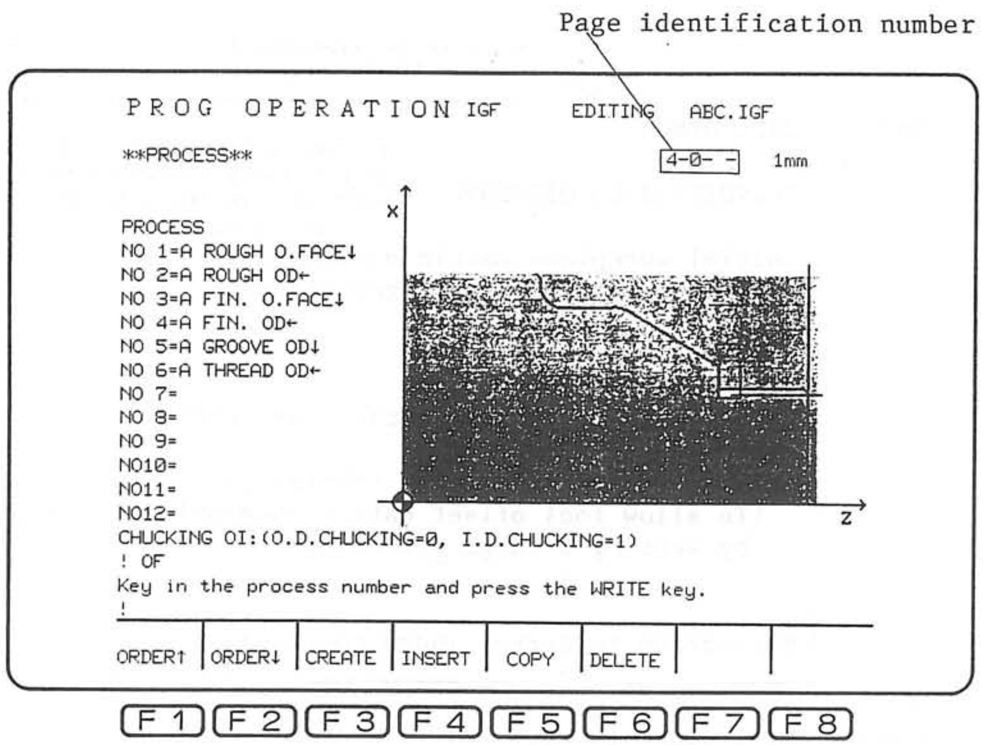
(3) The System variables used for tool animation designated in the gauging process are not included in NC programs.

Therefore, in the automatic operation mode graphic display, the tool form data (tool data setting mode) assigned for the turret position on which the sensor is installed, are displayed as animated. Therefore, set the proper data in the tool data setting mode.

SECTION 15 FUNDAMENTAL GRAPHIC EDIT OPERATION FLOW AND CURRENT OPERATION POSITION

Graphic edit operations are divided into a number of steps to allow the operator to enter data without complicated processes. The IGF system accordingly provides a number of display pages.

One potential disadvantage to this system is the possibility of the operator losing track of where he is in the data entry operation. To avoid any possible confusion, a page identification number has been included at the upper right of each page in the graphic edit function to allow the operator to keep track of precisely which procedure he is conducting at any given time.



Operation processes of the IGF graphic editing have four hierarchy levels as shown in the figures on the following pages, Graphic Edit Operations - Operation Flow.

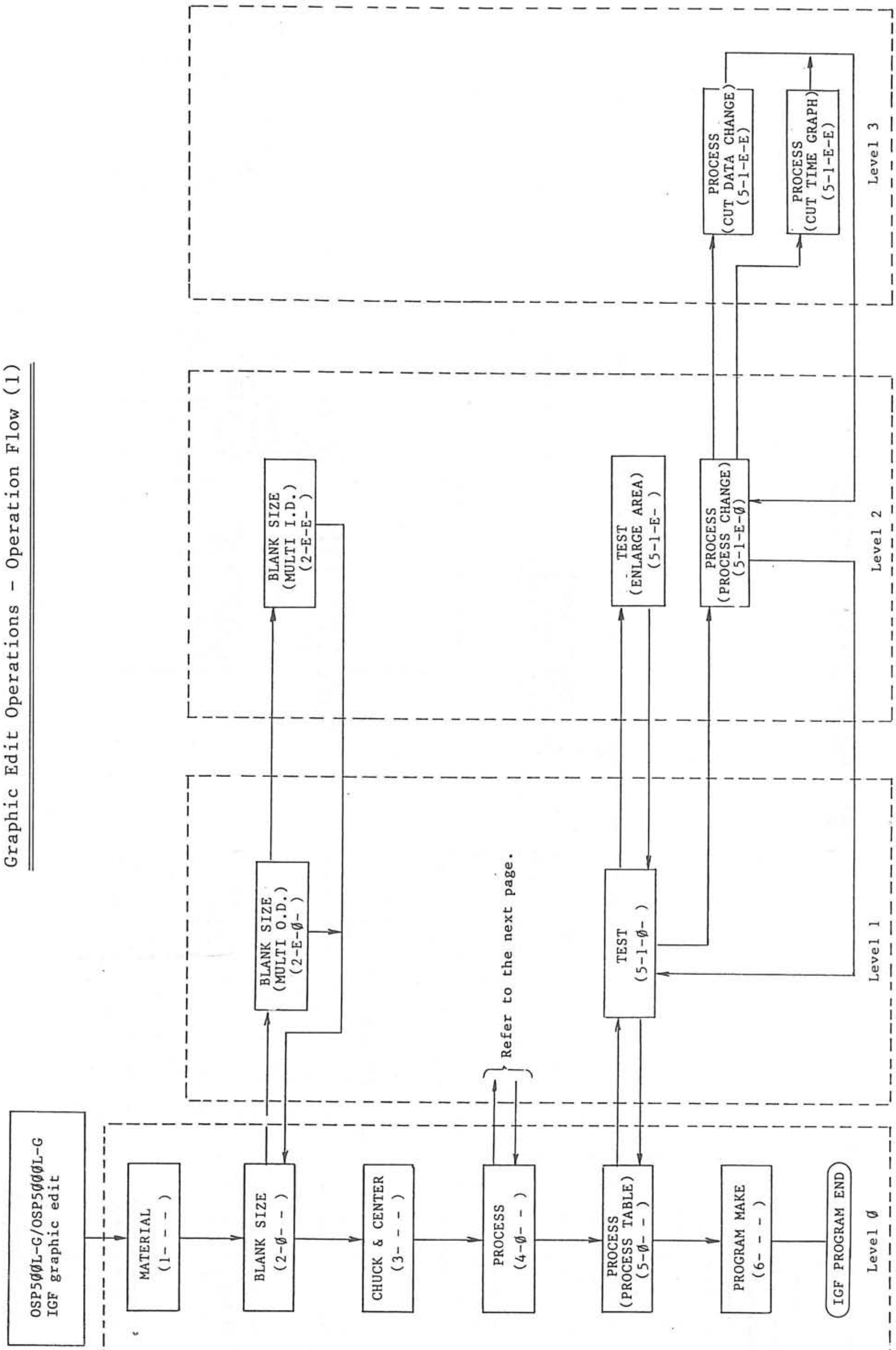
Each digit of the page identification numbers has following function.

9 - 9 - 9 - 9

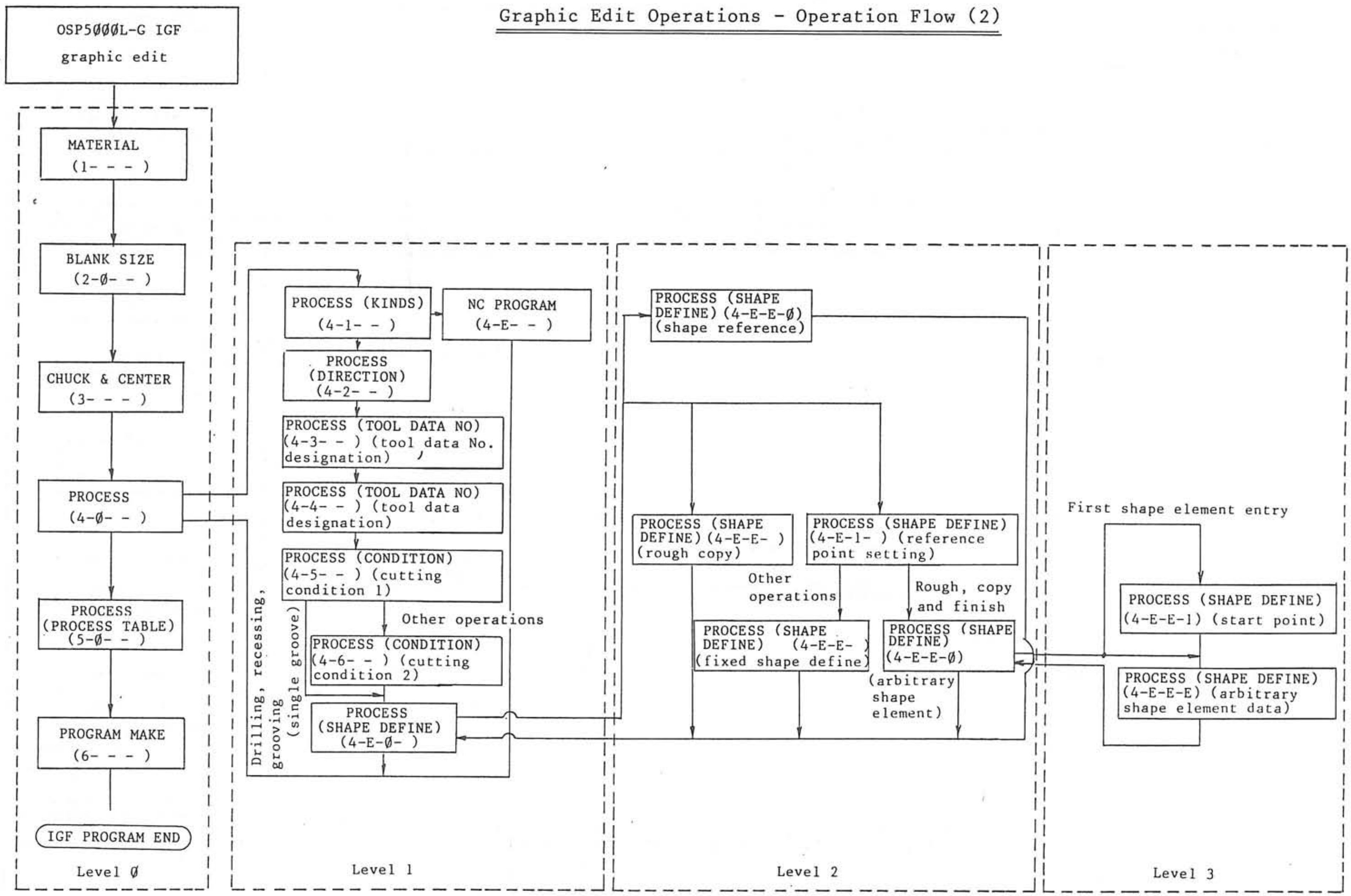
- Identification number in level 3 (1, E and blank)
- Identification number in level 2 (1, E and blank)
- Identification number in level 1 (1 - 6, E and blank)
- Identification number in level Ø (1 - 4)

In this section, the page identification number is simply called the page number.

Graphic Edit Operations - Operation Flow (1)



Graphic Edit Operations - Operation Flow (2)



The number-string in the parentheses in the boxes of the charts on the preceding pages correspond to the page numbers which are displayed on each display page.

The page number provides a means to recognize whether the function has another hierarchy level or not. That is, if the number corresponding to the next hierarchy level is no provided (or empty), it means that the function has no next level and if it is "Ø", the function has another level.

If the number corresponding to the present level is "E", the operation can be returned to the level one level higher than the present level by pressing the function key [END]. Note that this return operation is possible only when the necessary data has all been entered.

An example operation flow is outlined below assuming rough cutting operations. (Flow after the start of graphic-edit)

Processing	Operations	Page No.	Level
MATERIAL ⇩	Designate the blank material in the coded number.	1- - -	Ø
BLANK SIZE ⇩	Enter blank length(s) and diameter(s) (multiple OD/ID).	2-Ø- -	Ø
CHUCK & CENTER ⇩	Enter chuck and tailstock spindle dimensions.	3- - -	Ø
PROCESS ⇩	Process table is displayed. For designating processes newly, press function key [CREATE].	4-Ø- -	Ø
PROCESS (KINDS) ⇩	Select the desired process from menu. 1=ROUGH, for example.	4-1- -	1
PROCESS (DIRECTION) ⇩	Select cutting direction from menu. 1=OD ←, for example	4-2- -	1
PROCESS (TOOL DATA NO) ⇩	Designate the tool number of the tool to be used for the specified cutting.	4-3- -	1
PROCESS (TOOL DATA NO)	Check the tool data. Make corrections if necessary.	4-4- -	1

Processing	Operations	Page No.	Level
PROCESS (CONDITION) ⇩	Check the cutting conditions determined automatically and displayed. Make corrections if necessary.	4-5- -	1
PROCESS (CONDITION) ⇩	Check the finishing conditions determined automatically and displayed. Make corrections if necessary.	4-6- -	1
PROCESS (SHAPE DEFINE) ⇩	Determine whether the data is to be input newly or the data already stored is edited. Select "CREATE", for instance.	4-E-Ø-	1
PROCESS (SHAPE DEFINE) ⇩	Entry of Cycle Reference Point Check the reference point data determined automatically and displayed. Edit it if required.	4-E-1-	2
PROCESS (SHAPE DEFINE) ⇩	Definition of Shape Enter the shape using the function keys representing shape elements.	4-E-E-Ø	2
PROCESS (SHAPE DEFINE) ⇩	Entry of Start Point Enter the start point data. The prompt START PT is given only for the first shape element.	4-E-E-1	3
PROCESS (SHAPE DEFINE)	Entry of Shape Element Data Enter the data for defining the shape element selected.	4-E-E-E	3

Processing	Operations	Page No.	Level
PROCESS (SHAPE DEFINE) ⇩	Definition of Shape Enter the shape using the function keys representing shape elements.	4-E-E-Ø	2
PROCESS (SHAPE DEFINE) ⇩	Entry of Shape Element Data Enter the data for defining the shape element selected. After the completion of entry of shape element definition data for the shape element selected, the step will return to the shape element entry.	4-E-E-E	3
PROCESS (SHAPE DEFINE) ⇩	After the completion of entry of shape defining data for all shape elements selected, the page number should be as indicated at the right. Press the [QUIT] key.	4-E-E-Ø	2
PROCESS (SHAPE DEFINE) ⇩	Press the [QUIT] key.	4-E-Ø-	1
PROCESS ⇩	Press the [ORDER↓] key.	4-Ø- -	Ø
PROCESS ⇩	Process Table The Process Table is displayed. Carry out PROCESS TEST.	5-Ø- -	Ø
PROGRAM MAKE	Enter a program name and make an NC program from the IGF data. OKUMA IGF	6- - -	Ø

SECTION 16 ANIMATION DATA

16-1. SYSTEM VARIABLES FOR ANIMATION DISPLAY

The tool shape data, chuck barrier data and tailstock barrier data can be commanded in the same manner as conventional part program using the system variables. Usually, with the IGF system, the tool shape data, the chuck barrier data and the tailstock barrier data are automatically generated using the system variables from the tool data selected and the definition on the CHUCK & TAILSTOCK page.

16-1-1. Tool Shape Definition

The seven system variables indicated below are used for defining the tool shape.

Variable Name	Contents	Data Size
VTLIN[I]	Tool classification code	1 - 26
VTLFN[I]	Tool shape code	1 - 4
VTLA1[I]	Tool angle	Ø - 36Ø.ØØØ
VTLA2[I]	Edge angle	-36Ø.ØØØ - 36Ø.ØØØ
VTLL[I]	Holder length/Sticking out length/Drill length	Ø - 9999.999
VTLD[I]	Holder diameter/Drill diameter	Ø - 9999.999
VTLW[I]	Tool width	Ø - 9999.999

The index [I] of the system variable is designated by the tool number.

The tool number indicates the station number of the turret.

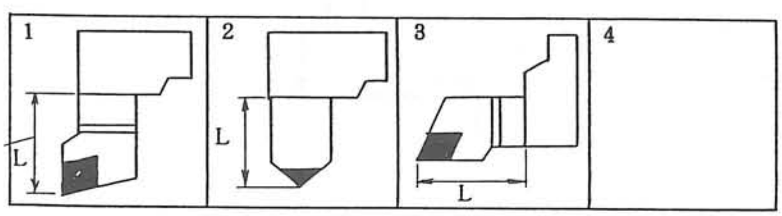
For the tool life management specification, the system variable name is assigned in the format VGR**[I]. The index [I] is the tool group number.

Tool Classification Code:

1 = ROUGH OD ←
2 = ROUGH ID ←
3 = ROUGH FACE ↓
4 = ROUGH OD →
5 = ROUGH ID →
6 = ROUGH FACE ↑
7 = FINISH OD ←
8 = FINISH ID ←
9 = FINISH FACE ↓
10 = FINISH OD →
11 = FINISH ID →
12 = FINISH FACE ↑
13 = THREAD OD ←
14 = THREAD ID ←
15 = THREAD FACE ↓
16 = THREAD OD →
17 = THREAD ID →
18 = THREAD FACE ↑
19 = GROOVE OD ↓
20 = GROOVE ID ↑
21 = GROOVE FACE ←
22 = DRILL HSS
23 = DRILL CARBIDE
24 = DRILL CENTER
25 = RECESS OD ↙
26 = RECESS ID ↖

Tool Shape Code: 1, 2, 3, 4 — Tool shape number which corresponds to the tool shape usable for the tool selected by the tool classification code.

Sticking out length



(ROUGH OD ←, FINISH OD ←)

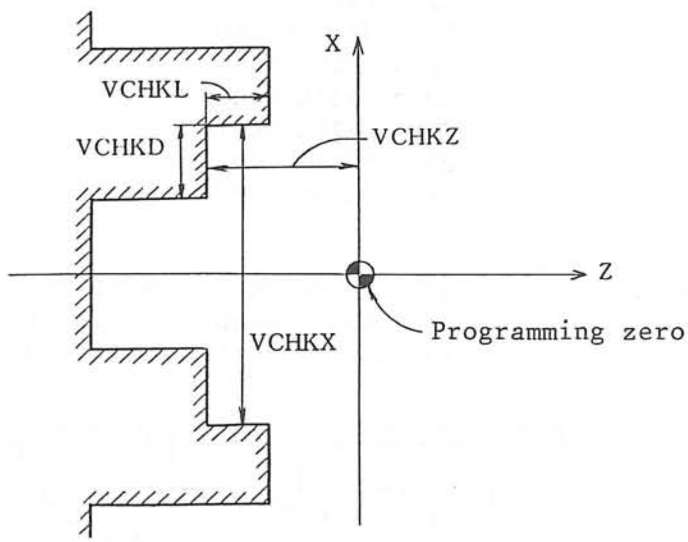
Example: ROUGH OD ←

```
TOOL NO.        TN=2
TOOL ANGLE      A1=80°
EDGE ANGLE      A2= 3°
STICKING OUT    L=40 mm

VTLIN[2] = 1
VTLFN[2] = 1
VTLA1[2] = 80.000
VTLA2[2] =  3.000
VTLL[2]  = 40.000
```

16-1-2. Chuck Barrier Definition

The four system variables indicated below are used for defining the chuck barrier.

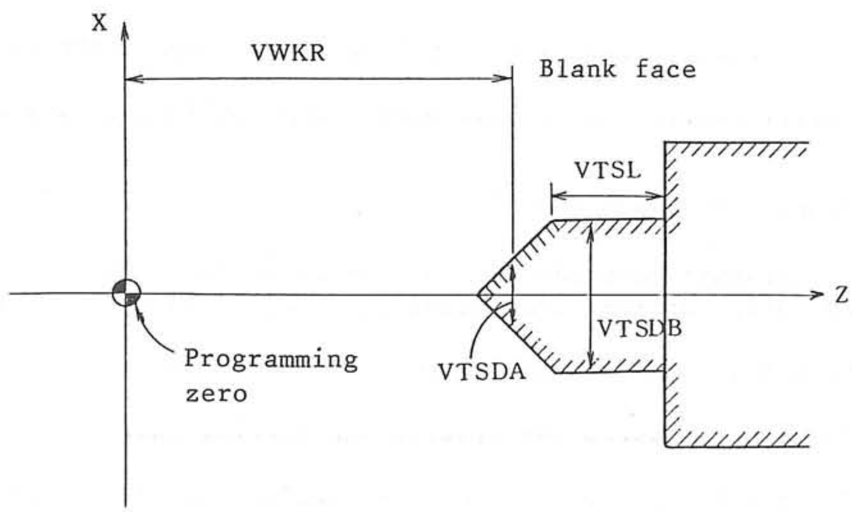
Variable Name	Contents	Data Size
VCHKL	Chuck jaw length	Ø - 9999.999
VCHKD	Step on chuck jaw	Ø - 9999.999
VCHKX	Gripping diameter	-9999.999 - 9999.999
VCHKZ	Distance from programming zero	-9999.999 - 9999.999



16-1-3. Tailstock Barrier Definition

The four system variables indicated below are used for defining the tailstock barrier.

Variable Name	Contents	Data Size
VWKR	Blank face position	-9999.999 - 9999.999
VTSL	Sticking out length of tailstock spindle	Ø - 9999.999
VTSDA	Center hole diameter	Ø - 9999.999
VTSDB	Tailstock spindle diameter	Ø - 9999.999



16-2. DRAWING BLANK SHAPE

The user graphic commands (UGC) are provided so that arbitrary blank shapes may be drawn on the graphic CRT. With the IGF system, the blank shape is automatically generated using the user graphic commands by the entry of the blank data through the interactive operations.

16-2-1. Definition Format

The format used for blank shape drawing is indicated below.

DEF WORK .... Declaration of blank definition

A group of user graphic commands used for drawing blank shape

END .... End of blank shape definition

DRAW .... Blank shape defined is drawn

16-2-2. User Graphic Commands (UGC)

The user graphic commands are largely classified into four types such as drawing point setting, line drawing, circle drawing and painting.

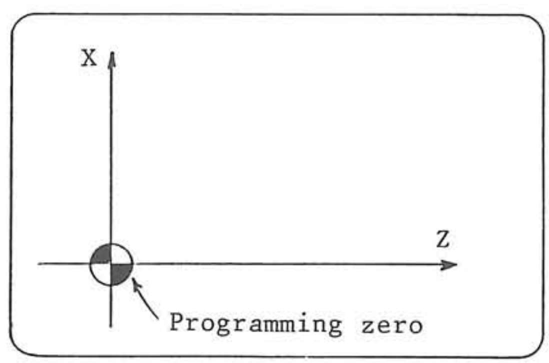
The explanation below provides the format of the representative commands.

(1) Coordinate System for Drawing and Setting Unit

The coordinate system used for drawing the blank shape is the Z-X coordinate system having the programming zero as the origin.

The unit system used is:

1 mm for metric system, and Ø.1 inches for inch system



(2) Setting Drawing Point

```
POINT [ZØ, XØ]
```

Note: The command underlined may be used instead of fully giving command characters. POINT, for instance, can be commanded by the underlined characters PO.

The POINT command simply sets the starting point (ZØ, XØ) for the drawing and no actual drawing operation occurs.

This sets the last reference point (LP) for the drawing.

(3) Drawing Straight Line

```
LINE [Z1, X1]<,line-code>
```

A straight line is drawn from the last reference point (LP) of the drawing up to the commanded end point. The end point is referenced to the LP. That is, the command must be given in a relative value. After the execution of the command, the LP is established at the end point of this command.

Line code:
- Ø = Solid line
- 1 = Dotted line
- 2 = Short dashes line
- 3 = Long dashes line
- 4 = Alternate short and long dashes line
- 5 = Alternate long and two short dashes line
- 6 = No axis display
- 7 = Erasing

Default is "Ø (solid line)".

(4) Circle Drawing

```
CIRCLE [Z1, X1], [Zc, Xc] <,rotation-direction><,line-code>
```

An arc is drawn in the following conditions:

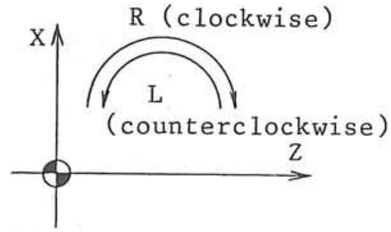
Starting point:	LP
End point:	(Z1, X1)
Center:	(Zc, Xc)

The coordinates of the end point and the center are referenced to the LP. After the execution of the command, the LP is established at the end point of this command.

Rotation Direction:

R: Clockwise
L: Counterclockwise

Default is "R (clockwise)".



Line code: Eight codes, Ø through 7

(5) Painting

PAINT (or PF) <number-of-vertexes><,vertex-coordinates> <,tile-pattern>

The PAINT command is used for painting triangles, rectangles and circles defined on the absolute coordinate system.

Number of Vertexes:

Ø: Circles or sectors
3: Triangles
4: Rectangles

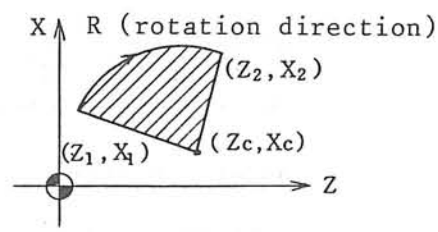
Vertex Coordinates:

The designation of coordinates depends on the number of vertexes specified.

<u>Number of vertexes = Ø</u>

[Zc, Xc], [Z1, X1], [Z2, X2] <,rotation-direction>

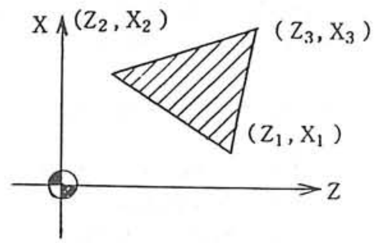
Zc, Xc: Center
Z1, X1: Start point
Z2, X2: End point



Number of vertexes = 3

[Z1, X1], [Z2, X2], [Z3, X3]

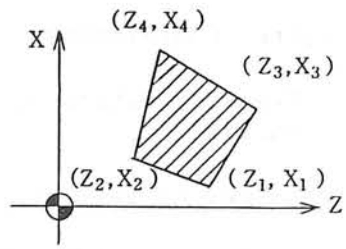
(X1 ≦ X2 ≦ X3)



Number of vertexes = 4

[Z1, X1], [Z2, X2], [Z3, X3], [Z4, X4]

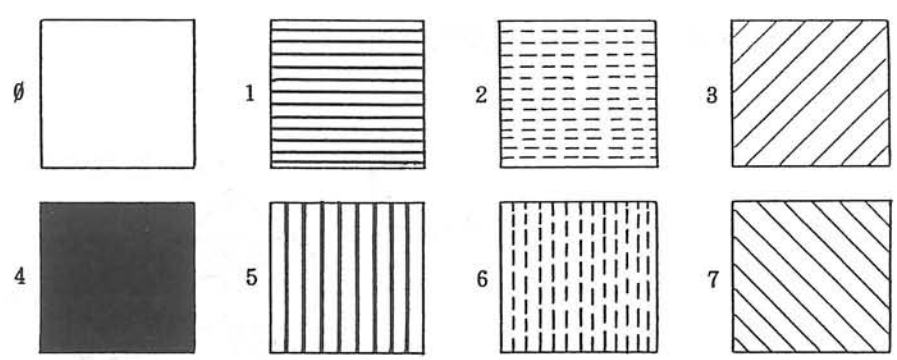
(X1 ≦ X2 ≦ X3 ≦ X4)



Note that the coordinates of vertexes must be given in absolute values.

Tile Pattern:

Ø: No drawing (all dots off)
1: All dots painted; on every other horizontal line
2: Dots alternately on and off; on every other horizontal line
3: Oblique line; from upper right to lower left
4: All dots painted
5: All dots painted; on every other vertical line
6: Dots alternately on and off; on every other vertical line
7: Oblique line; from upper left to lower right



Default is "4".

```
PAINTI <number-of-vertexes><,reference-point-coordiantes>
       <,vertex-coordinates><,tile-pattern>
```

The PAINTI command is used for painting triangle, rectangles and circles defined on the incremental coordinate system in reference to the reference point specified.

Number of Vertexes:

- Ø: Circles or sectors
- 3: Triangles
- 4: Rectangles

Reference Point Coordinates:

The coordinates of the reference point for defining a shape; [ZØ, XØ]

The values must be given in absolute values.

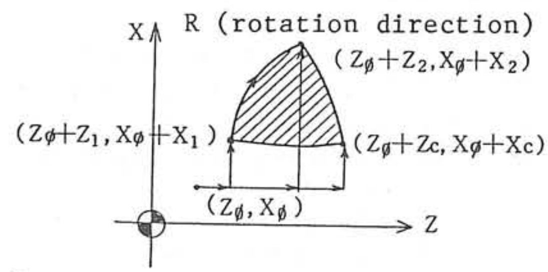
Vertex Coordinates:

The designation of coordinates depends on the number of vertexes specified.

Number of vertexes = Ø

[Zc, Xc], [Z1, X1], [Z2, X2] ,<rotation-direction>

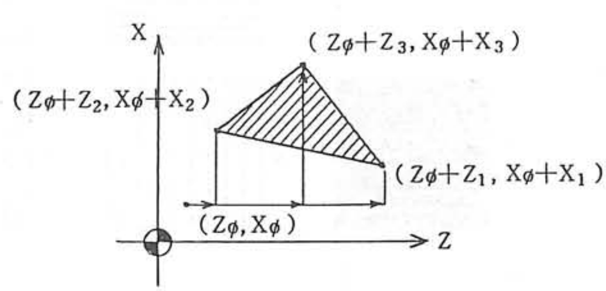
Zc, Xc: Center
Z1, X1: Start point
Z2, X2: End point



Number of vertexes = 3

[Z1, X1], [Z2, X2], [Z3, X3]

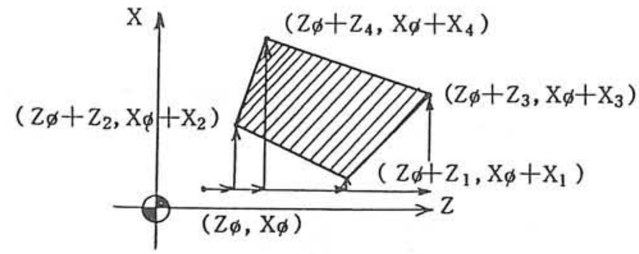
(X1 ≤ X2 ≤ X3)



Number of vertexes = 4

[Z1, X1], [Z2, X2], [Z3, X3], [Z4, X4]

(X1 ≦ X2 ≦ X3 ≦ X4)



Note that the coordinates of vertexes must be given in incremental values.

Tile Pattern: Eight patterns, Ø through 7

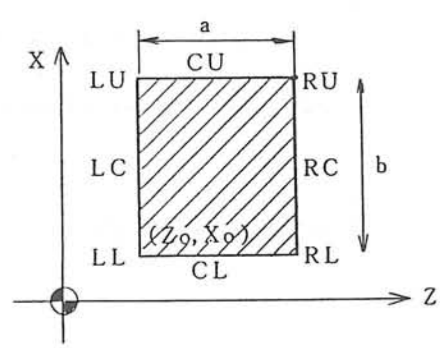
```
PAINTS <reference-point-position>,[ZØ, XØ],[a, b]
       <,tile-pattern>
```

The PAINTS command is used for painting a rectangle having lengths of a and b.

Reference Point Position:

This specifies the position of the reference point on the rectangle being painted.

LL: Left lower
LC: Left center
LU: Left upper
CU: Center upper
RU: Right upper
RC: Right center
RL: Right lower
CL: Center lower



Reference Position [ZØ, XØ]:

The coordinates must be specified in absolute values.

Tile Pattern: Eight patterns, Ø through 7

PAINTP <reference-point-position>,[ZØ, XØ],[a, b]
<,angle-code><,tile-pattern>

The PAINTP command is used for painting rectangles and triangles.

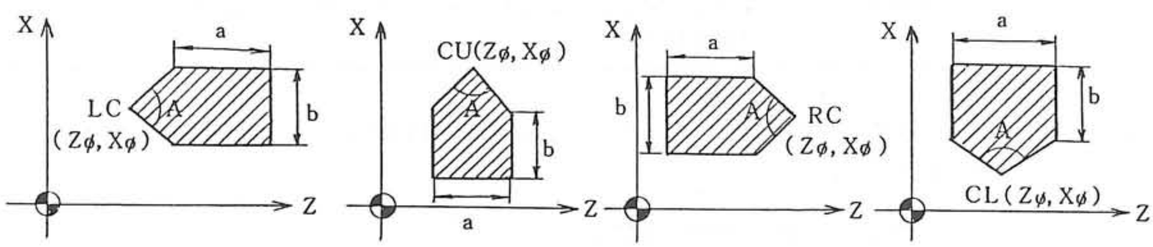
Reference Point Position:

This specifies the position of the reference point on the rectangle and triangle being painted.

LC: Left center
CU: Center upper
RC: Right center
CL: Center lower

Reference Position [ZØ, XØ]:

The coordinates must be specified in absolute values.



Angle Code:

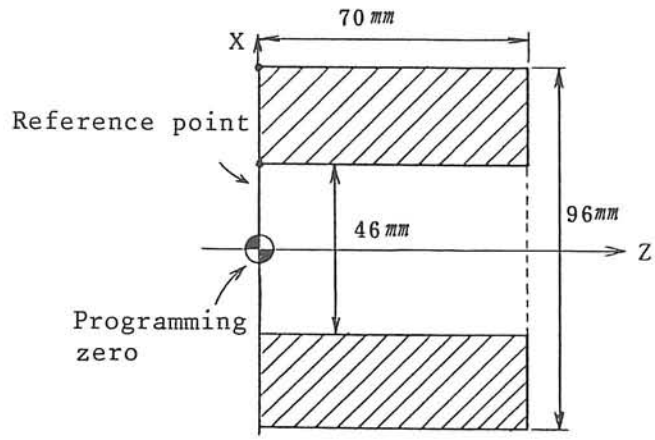
Ø: Angle A of triangle is 3Ø°
1: Angle A of triangle is 6Ø°
2: Angle A of triangle is 12Ø°

Default is "Ø = 3Ø°".

Tile Pattern: Eight patterns, Ø through 7

(6) Example of Painting

This item explains the procedure to paint the blank shape indicated below.



This painting operation uses the PAINTS command.

```
Reference point [ZØ, XØ] = [Ø, Ø]
Dimensions      [a, b]   = [7Ø, 96]
```

```
PAINTS LC,[Ø, Ø],[7Ø, 96],4
```

The opening in the blank should be defined using the tile pattern "Ø" which represents all dots off.

```
Reference point [ZØ, XØ] = [Ø, Ø]
Dimensions      [a, b]   = [7Ø, 46]
```

```
PAINTS LC,[Ø, Ø],[7Ø, 46],Ø
```

Therefore, the painting is programmed as indicated below:

```
DEF WORK
PAINTS LC,[Ø, Ø],[7Ø, 96],4
PAINTS LC,[Ø, Ø],[7Ø, 46],Ø
END
DRAW
```

Using simple commands, the same painting is programmed as indicated below:

```
DEF WORK
PS LC,[Ø, Ø],[7Ø, 96]
PS LC,[Ø, Ø],[7Ø, 46],Ø
END
DRAW
```

Interactive Graphic MDI Function

IGF-L3

OSP500L-G/OSP5000L-G CNC SYSTEMS

ERROR LIST

(1st Edition)

OKUMA

Your single source for machine & control

Publication No. CB002 2908-E
(April 1987)

PREFACE

The CRT screen displays an error message at the lower left area on it when a wrong operation is attempted. This manual is prepared so that an operator can check which operation was erroneous by referring to this list.

The error messages are assigned with four digit error numbers, which can be used to consult this error message table.

IGF ERROR MESSAGES

Error No.	Error Name
28Ø1	Process number designate error
28Ø2	Process number over error
28Ø3	IGF program overflow error
28Ø4	Roughing copy command error
28Ø5	No data setting error
28Ø6	IGF file read error
28Ø7	Input data overflow (+/-) error
28Ø8	Material name input error
28Ø9	Numerical data error
281Ø	Numerical digit error
2811	Decimal point error
2812	IGF graphic panel command data table error
2813	Pattern edit buffer over error
2814	Pattern designate calculation error
2815	Pattern package calculation error
2816	File & machine unit system mismatch error
2817	Angle command error
2818	No depth of cut command error
2819	Cutting start point error
282Ø	IGF data capacity over error
2821	Program name error
2822	File name error
2823	Program read buffer over error
2824	Tool pattern number no entry error
2825	Spindle rotate direction unmatched error
2826	Process combination impossible error
2827	Data set error
2828	Process change error
2829	Not found main program name error
283Ø	Program end code error
2831	Program record read error
2832	Program buffer overflow error
2833	M machining X, Y data error
2834	Program making calculation error
2835	Face contour generating data error
2836	V-groove angle data error
2837	IGF file version no. not same error
2838	IGF file machine code not same error
2839	IGF file spec. code not same error
284Ø	Undefined auxiliary shape error
2841	Undefined gauging object process error
2842	Unmatched gauging object process error
2843	Calculating expression error
2844	Shape reference command error
2845	Turret No. error

Error No.	Error Name
29ØØ	Expression: right part error
29Ø1	Expression: calculation error
29Ø2	Expression: syntax error
29Ø3	Expression: buffer over error
29Ø4	Program factor: G-code error
29Ø5	Program factor: M-code error
29Ø6	Program factor: common var error
29Ø7	Program factor: sequence name error
29Ø8	Program factor: buffer over error
29Ø9	Program factor: illegal char error
291Ø	Program factor: program name error
2911	Program syntax: G-code error
2912	Program syntax: G13/G14 change error
2913	Program syntax: M-code error
2914	Program syntax: equal error
2915	Program syntax: left part error
2916	Program syntax: special G-code error
2917	Program syntax: turret code error
2918	Program syntax: program end error
2919	Data word: 'F' error
292Ø	Data word: 'F' or 'E' error
2921	Data word: 'I' error
2922	Data word: 'J' error
2923	Data word: 'K' error
2924	Data word: 'L' error
2925	Data word: 'P' error
2926	Data word: 'S' error
2927	Data word: no 'S' error
2928	Data word: 'T' error
2929	Data word: 'X' error
293Ø	Data word: 'X'.'Z' error
2931	Data word: 'Z' error
2932	Data word: arc cal. error
2933	Data word: angle error
2934	Data word: radius error
2935	Data word: C command error
2936	Multi cycle: B illegal order error
2937	Multi cycle: D illegal order error
2938	Multi cycle: F illegal order error
2939	Multi cycle: H illegal order error
294Ø	Multi cycle: H-U(W) less than D(M73) error
2941	Multi cycle: parameter I, K over error
2942	Multi cycle: I, K illegal order error
2943	Multi cycle: L illegal order error
2944	Multi cycle: entry in LAP error
2945	Multi cycle: U(W) illegal order error

Error No.	Error Name
2946	Multi cycle: U(W) greater than H error
2947	Multi cycle: X, Z illegal order error
2948	Multi cycle: angle error
2949	Multi cycle: tool offset error
295Ø	Multi cycle: cycle start point error
2951	Multi cycle: entry in NOSE-R error
2952	Multi cycle: width error
2953	Chamfering: GØ1 mode error
2954	Chamfering: parameter L over error
2955	Chamfering: L illegal order error
2956	Chamfering: X, Z illegal order error
2957	LAP: B illegal order error
2958	LAP: D illegal order error
2959	LAP: DA(DB) illegal order error
296Ø	LAP: H illegal order error
2961	LAP: H-U(W) less than D(M73) error
2962	LAP: U(W) illegal order error
2963	LAP: U(W) greater than H error
2964	LAP: XA(ZA), XB(ZB) illegal order error
2965	LAP: calculation error
2966	LAP: a number of down stair over error
2967	LAP: entry in LAP error
2968	LAP: sequence name error
2969	LAP: LAP control error
297Ø	LAP: G-code error
2971	LAP: entry in NOSE-R error
2972	NOSE-R comp.: calculation error
2973	NOSE-R comp.: cancel impossible error
2974	NOSE-R comp.: no cross point error
2975	NOSE-R comp.: no spec. error
2976	NOSE-R comp.: start up impossible error
2977	NOSE-R comp.: thread cycle error
2978	Tool life control: no spec. error
2979	Tool life control: tool group error
298Ø	Tool life control: no T-entry error
2981	Tool life control: tool offset group error
2982	Tool life control: no T-offset error
2983	Cannot IGF-convert G82 (N****) error
2984	Cannot IGF-convert G32 (N****) error
2985	Cannot IGF-convert G72 (N****) error
2986	Cannot IGF-convert B command (N****) error
2987	Cannot IGF-convert L command (N****) error
2988	Cannot IGF-convert K command (N****) error
299Ø	Cannot IGF-convert TG command (N****) error
2991	Cannot IGF-convert OG command (N****) error
2993	LAP: NOSE-R not cancelled error

Error No.	Error Name
2994	NOSE-R comp.: NOSE-R > circle-R error
2999	Output buffer over (N****) error
3ØØ5	Data word: SB error
3ØØ6	Fixed cycle: no spec. error
3ØØ7	Fixed cycle: C error
3ØØ8	Fixed cycle: I, K error
3ØØ9	Fixed cycle: Q error
3Ø1Ø	Fixed cycle: F error
3Ø11	Fixed cycle: L error
3Ø12	Fixed cycle: D error
3Ø13	Fixed cycle: X, Z error
3Ø14	Fixed cycle: SA error
3Ø15	Fixed cycle: feed G94 error
3Ø17	Fixed cycle: thread cycle error
3Ø18	Data word: QA error
3Ø19	Data word: X, Y command error
3Ø2Ø	Data word: incremental error
3Ø21	Data word: no X, Y error
3Ø22	Data word: 'Y' error
3Ø23	Data word: distance cal error
3Ø24	Data word: 'R' error

28Ø1 Process number designate error

The process number specified in the process operation (creation, copy, deletion, etc.) in the graphic edit is not correct.

28Ø2 Process number over error

The number of processes exceeded the allowable size in the process operation (creation, insertion, etc.).

28Ø3 IGF program over error

During the IGF graphic editing, process data has exceeded the memory capacity and the data cannot be stored in the memory.

28Ø4 Roughing copy command error

An attempt to make ROUGH COPY operation in other than the finishing process definition

28Ø5 No data setting error

An attempt to advance the process by pressing the function key [ORDER ↑] although required data has not all been entered

28Ø6 IGF file read error

Attribute or the size of the IGF data file designated in the IGF graphic edit process or in the program making process is improper.

28Ø7 Input data overflow (+/-) error

Input data overflow

28Ø8 Material name input error

In IGF material data setting operation, improper data setting for material name was attempted.

28Ø9 Numerical data error

281Ø Numerical digit error

The number of digits of the data to be entered is improper.

2811 Decimal point error

An attempt to enter the data with decimal point for parameters which accept only integer data

2812 IGF graphic panel command data table error

The graphic panel command data table is incorrect.

2813 Pattern edit buffer over error

The number of shape elements used for defining the rough, copy and finish shape exceeded the maximum capacity of 48.

2814 Pattern designate calculation error

An operation error occurred during the execution of the rough, copy and finish shape definition.

2815 Pattern package designate calculation error

An error occurred in the graphic calculation package during the definition of the rough, copy and finish shape.

2816 File & machine unit system mismatch error

Mismatch between the unit system of the IGF file designated in the IGF graphic edit process or in the program making process and the machine unit system.

2817 Angle command error

NC program cannot be compiled since the angle command in the thread cutting process is improper.

2818 No depth of cut command error

NC program cannot be compiled since step infeed amount for pecking cycle or tool retraction amount is not set for the grooving or drilling process.

2819 Cutting start point error

Cutting starting point is set on the reference point for the rough, copy and finish processes.

2820 IGF data capacity over error

The size of the NC program to be made using the graphic edit function is too large.

2821 Program name error

Program name does not meet the following requirements.

a) Begins with character "O"; within five characters.

b) Spaces between characters not allowed.

c) If the character following the character "O" is an alphabet, then four characters following "O" may be alphanumeric.

d) If the character following the character "O" is a figure, then four characters following "O" must be figures.

2822 File name error

File name does not meet the following requirements.

a) Extension must be .MIN, .SUB or .SSB. (.MIN is omissible.)

b) If the extension is .SSB or .SUB, a program name must be specified.

2823 Program read buffer over error

During the IGF graphic editing, process data has exceeded the memory capacity and the data cannot be stored in the memory.

The data size entered is approximately calculated using the following formula.

$$650 + \{300 \times \text{(number of processes)}\} + \{60 \times \text{(number of shape elements in total processes)}\}$$

The sum above must not exceed 12,ØØØ (characters).

2824 Tool pattern number no entry error

Tool pattern number is not entered.

2825 Spindle rotate direction unmatched error

2826 Process combination impossible error

2827 Data set error

2828 Process change error

2829 Not found main program name error

283Ø Program end code error

2831 Program record read error

2832 Program buffer overflow error

2833 M machining X, Y data error

Part programming cannot be made since X and Y values for defining contour for multi-machining are incorrect.

2834 Program making calculation error

Calculation error found during part program creation.

2835 Face contour generating data error

Shape data defined for face contour generating is not correct and program cannot be created.

2836 V-groove angle data error

An incosistency was found in the V-groove angle data and program cannot be created.

2837 IGF file version no. not same error

The IGF file being loaded in the graphic edit mode does not match the current version.

2838 IGF file machine code not same error

Machine code of the IGF file being loaded in the graphic edit mode does not match the machine for which the system is set.

Code : ********

	Machine Type		Machine Type
1ØØØØØØ	LC1Ø	1ØØØØ	LB6
2ØØØØØØ	LC2Ø	2ØØØØ	LB8
4ØØØØØØ	LC3Ø	4ØØØØ	LB1Ø
8ØØØØØØ	LC4Ø	8ØØØØ	LB15
1ØØØØØØØ	LC5Ø	1ØØØØØ	LP15
2ØØØØØØØ	LS3Ø	2ØØØØØ	LR2Ø
4ØØØØØØØ	LH35	4ØØØØØ	LR3Ø
8ØØØØØØØ	LH55	8ØØØØØ	LR4Ø

2839 IGF file spec. code not same error

Mismatch of IGF file specification

Code :

	Specification
None ...	1-saddle
1	2-saddle
2	Two-turret model (Mirror image)
4	Multi-machining model
8	ATC

284Ø Undefined auxiliary shape error

An auxiliary shape function key has been pressed even though no auxiliary shape data has been designated.

2841 Undefined gauging object process error

The process for which gauging is to be made was not present when the gauging process is being made.

2842 Unmatched gauging object process error

The process selected is the one which was not selectable as the gauging process, or the process selected did not contain the machined surface meeting the designated gauging direction (OD, ID, FACE).

Code : 1 .... The process for which the gauging cycle cannot be conducted was selected as the object of the gauging process.

→ Change the process.

2 .... The process which does not have the machined surface meeting the designated gauging direction was selected as the gauging process.

→ Change the process or the gauging direction.

2843 Calculating expression error

An error was discovered in the calculating expression, or an error occurred during calculation.

Code : 1ØØ ..... Improper operator is used.
2ØØ ..... Calculation order is improper.
3ØØ ..... Calculation stack overflow
4ØØ ..... No "]" corresponding to "["
5ØØ ..... No "[" corresponding to "]"
6ØØ ..... Inconsistency in operand
7ØØ ..... Overflow of the number of digits of input data
8ØØ ..... More than decimal point (.) is used in input data.

9XX ..... Real number calculation error

XX: Status information
Bit Ø ... Overflow in converting into integer
Bit 1 ... Exponential underflow
Bit 2 ... Exponential overflow
Bit 3 ... Calculation of root of a negative number
Bit 4 ... Division by Ø
Bit 5 ... Angle overflow for TAN

AXX ..... Integer calculation error

XX: Status information
Bit Ø ... Overflow in addition
Bit 1 ... Overflow in subtraction
Bit 2 ... Overflow in multiplication
Bit 3 ... Underflow in multiplication
Bit 4 ... Overflow in division
Bit 5 ... Underflow in division

2844 Shape reference command error

An attempt was made to reference the shape in the process for which shape reference is not allowed, or the attempt to reference the shape was made although the shape has not been defined.

Code : 1 .... An attempt was made to reference the shape for which shape reference is not allowed. Shape reference is allowed for rough, copy, finish and creation face machining.
2 .... An attempt was made to reference the shape although shape has not been defined.

2845 Turret No. error

For LR15M-ATC, L/M designation of the turret position selected does not match the L/M designation of the tool data number.

29ØØ Expression: right part error

An illegal command is programmed in the right part of the expression.

Commands other than numerical data, input variables, system variables, common variables, local variables and extended address characters are programmed.

Index : TURRET
Character-string: None
Code : Command factor classification code (See Table 1 for details.)

29Ø1 Expression: calculation error

Calculation alarm

Index :	TURRET
Character-string:	None
Code :	XXYY

XX:

Bit Ø .... Overflow in addition/subtraction
Bit 1 .... Overflow in converting ABS data into integer
Bit 2 .... Conversion form BCD to BIN
Bit 3 .... Conversion form BIN to BCD
Bit 4 .... DROUND, DFIX and DFUP command were programmed in other than mm (inch) unit system.

YY ...... Floating-point calculation error

YY:

Bit Ø .... Overflow in converting into integer
Bit 1 .... Exponential underflow
Bit 2 .... Exponential overflow
Bit 3 .... Calculation of root of a negative number
Bit 4 .... Division by Ø
Bit 5 .... Angle overflow

29Ø2 Expression: syntax

Syntax error of expression

Index	:	TURRET
Character-string	:	None
Code	:	1........ Calculation of subscript expression is intended within calculation of subscript expression.
		2 .... No left bracket "[" at the beginning of the subscript expression
		3 .... More than two subscript expressions
		4 .... The number of the left bracket "[" and that of the right bracket "]" do not match.
		5 .... The number of operators and their handling elements do not match.
		6 .... The sequence terminates within the expression.
		7 .... There are more than one solution.

29Ø3 Expression: buffer over

The number of expressions too many, making calculation impossible.

Index	:	TURRET
Character-string	:	None
Code	:	1 .... Overflow of operand stack in calculation of subscript expressions and operation expressions (more than 16)
		2 .... Overflow of operator data stack in calculation of operation expressions (more than 8)

29Ø4 Program bad direct: G-code

Illegal G code

Numerical value greater than 199 or less than Ø is assigned to the address character G ($Ø \leqq G \leqq 199$).

Index	:	TURRET
Character-string	:	None
Code	:	Hexadecimal number of the programmed numerical value

29Ø5 Program factor: M-code error

Illegal M code

Numerical value greater than 253 or less than Ø is assigned to the address character M ($Ø \leqq M \leqq 253$).

Index	:	TURRET
Character-string	:	None
Code	:	Hexadecimal number of the programmed numerical value

29Ø6 Program factor: common var. error

Wrong common variable designation

Index	:	TURRET
Character-string	:	None
Code	:	None .... Characters other than alphanumerics are programmed following address character V, or VØ is programmed.
		Other ... Variable number other than $1 \leqq V \leqq 32$ and $9Ø1 \leqq V \leqq 932$ is programmed. Variable number other than hexadecimal number of the variable number is programmed.

29Ø7 Program factor: sequence name error

Sequence name contains characters other than alphanumerics or too many characters are used.

Index	:	TURRET
Character-string	:	None
Code	:	1 .... No character follows address character N, or characters other than alphanumerics follow it.
		2 .... The number of characters following address character N is more than four.

29Ø8 Program factor: buffer over error

The buffer register storing program factors is full.

Index	:	TURRET
Character-string	:	None
Code	:	1 .... More than 127 factor classification codes and factor parameters are programmed. 2 .... More than 64 factor data are programmed. 3 .... Move range of factor classification code and/or factor parameter stack is wrong. 4 .... Move range of factor data stack is wrong (this error does not occur usually.)

29Ø9 Program factor: illegal char error

Illegal symbols are programmed.

Programmable symbols are "]", "[", "=", "*", "/", "+", "-", ",", DEL, BS, CR, HT and SP.

Index	:	TURRET
Character-string	:	None
Code	:	Hexadecimal number of ASCII code of the programmed symbol

291Ø Program factor: program name error

Program name contains characters other than alphanumerics or it contains too many characters.

Index	:	TURRET
Character-string	:	None
Code	:	1 .... No character follows address character O, or characters other than alphanumerics follow it. 2 .... The number of characters following address character O is more than four.

2911 Program syntax: G-code error

G code not available with the selected specification is programmed.

Index	:	TURRET
Character-string	:	None
Code	:	Hexadecimal number of the programmed G code

2912 Program syntax: G13/G14 change error

On two-turret models, G13 and G14 mode is changed while in incremental programming mode, tool nose radius compensation mode, LAP mode, and constant cutting speed mode.

Index	:	TURRET
Character-string	:	None
Code	:	1 .... Changed in incremental programming mode. 2 .... Changed in tool nose radius compensation mode. 3 .... Changed in LAP mode. 4 .... Changed in constant cutting speed mode. 5 .... Changed in buffer reading for required angle chamfering operation.

2913 Program syntax: M-code error

M code not available with the selected specification is programmed.

Index	:	TURRET
Character-string	:	None
Code	:	Hexadecimal number of the programmed M code

2914 Program syntax: equal error

"=" (equal) sign is incorrectly used.
(G codes, M codes, extended address character, local variables, common variables, system variables, and I/O variables)

Index	:	TURRET
Character-string	:	None
Code	:	Classification code and parameter of the factor programmed at a position where "=" should be programmed. XXYY: XX .... Factor classification code (See Table 1 for details) YY .... Factor parameter (See Table 1 for details)

2915 Program syntax: left part error

Illegal command in the left part of the expression.

Index	:	TURRET (None with schedule program)
Character-string	:	None
Code	:	Schedule program:

Left part is not common variable (V1 through V32) or output variable in VSET sequence. (Factor classification code)

Main program, Subprogram:

The left part contains other than G codes, M codes, address characters, extended address characters, local variables, common variables, system variables, and output variables.

XXYY:

XX .... Factor classification code
(See Table 1 for details)
YY .... Factor parameter
(See Table 1 for details)

Table 1 Factor Classification Code/Factor Parameter

Factor Classification Code (XX)	Factor Parameter (YY)
Ø3 ...]	ØØ
Ø4 ... EOR	ØØ
Ø6 ... OR	ØØ
Ø8 ... AND	ØØ
ØA ... NOT	ØØ
ØC ... Relative operator	Ø1 ... LT Ø2 ... LE Ø3 ... EQ Ø4 ... NE Ø5 ... GT Ø6 ... GE
ØE ...	ØØ
1Ø ... Adding/subtracting operator	Ø1 ... + Ø2 ... -
12 ... Multiplying/dividing operator	Ø1 ... * Ø2 ... /
18 ... Function operator	Ø1 ... SIN Ø2 ... COS Ø3 ... TAN Ø4 ... ATAN Ø5 ... ATAN2 Ø6 ... SQRT Ø7 ... ABS Ø8 ... BIN Ø9 ... BCD ØA ... ROUND ØB ... FIX ØC ... FUP ØD ... DROUND ØE ... DFIX ØF ... DFUP 1Ø ... MOD
1A ... [	Ø
1C ... =	Ø
22 ... Numerical value	Position of decimal point

2916 Program sysntax: special G-code error

The G code which must be programmed directly after the sequence number is programmed in wrong position.

The internal constant table determined by the special G code is incorrect.

Index	:	TURRET
Character-string	:	None
Code	:	Hexadecimal number of the programmed G code

2917 Program syntax: turret code error

G13 and G14 commands are programmed on machines having only one turret.

Index	:	TURRET
Character-string	:	None
Code	:	Hexadecimal number of the programmed G code

2918 Program syntax: program end error

No program end code is programmed.

Index	:	TURRET
Character-string	:	None
Code	:	1 .... No program end code 2 .... After the symbol "(" is programmed, corresponding symbol ")" does not appear up to the end of the program.

2919 Data word: 'F' error

Numerical value of an F word in other than the GØ4 mode is either negative or zero.

When the numerical value of an F word in other than the GØ4 mode is converted into "Ø.Ø1 sec." unit, it does not satisfy the following inequality:

$0 < F \leq 99999999$

Index	:	TURRET
Character-string	:	None
Code	:	1 .... F value does not satisfy; $-99999999 \leq F \leq 99999999$ 2 .... F value is either negative or zero.

2920 Data word: 'F' or 'E' error

Illegal F or E command

When F or E command is converted into "μm/rev." or "Ø.1 mm/min." unit, the result of conversion does not satisfy the following inequality:

$-99999999 \leqq F \text{ (or E)} \leqq 99999999$

Overflow in calculation of the number of feed pulse in the G34 or the G35 mode

Index : TURRET
Character-string: None
Code : 1 .... F or E value does not satisfy; $-99999999 \leqq F \text{ (or E)} \leqq 99999999$
3 .... Overflow in calculation of feed pulse numbers

2921 Data word: 'I' error

Illegal I command

Numerical value of I word is not:

$-99999.999 \leqq I \leqq 99999.999$

Index : TURRET
Character-string: None
Code : 1 .... I word in circular arc commands
Other ... Hexadecimal number of I word in thread cutting fixed cycle
Hexadecimal number of I word in other than circular arc commands or thread cutting fixed cycle

2922 Data word: 'J' error

Illegal J command

Numerical value of J word is not:

$0 < J \leqq 99999.999$

Index : TURRET
Character-string: None
Code : 1 .... J value is negative
2 .... J value does not satisfy: $0 < J \leqq 99999.999$

2923 Data word: 'K' error

Illegal K command

Numerical value of K word is not:

$-99999.999 \leq K \leq 99999.999$

Index : TURRET
Character-string: None
Code : 1 .... K word in circular arc commands
Other ... Hexadecimal number of K word in thread cutting fixed cycle
Hexadecimal number of K word in other than circular arc commands or thread cutting fixed cycle

2924 Data word: 'L' error

Illegal L command

Numerical value of L word in circular interpolation mode is not:

$\emptyset < L \leq 99999.999$

The chamfering amount in thread cutting fixed cycle calculated from L and K (or I) commands is not Ø through 99999.999.

Numerical value of L word in a gauging cycle is not Ø through 99999.999.

In other modes, numerical value of L word is not:

$-99999.999 \leq L \leq 99999.999$

Index : TURRET
Character-string: None
Code : 1 .... In circular interpolation mode, L word does not satisfy;
$-99999.999 \leq L \leq 99999.999$
2 .... L value is negative in circular interpolation mode
Other ... Hexadecimal number of L value in other than circular interpolation mode

2925 Data word: 'P' error

Illegal P command

Numerical value of P word is not:

$-9999 \leqq P \leqq 9999$

Index	:	TURRET
Character-string	:	None
Code	:	Hexadecimal number of programmed P word

2926 Data word: 'S' error

Illegal S command

Numerical value of S is not:

$\emptyset \leqq S \leqq 9999$

Index	:	TURRET
Character-string	:	None
Code	:	Hexadecimal number of programmed S word

2927 Data word: no 'S' error

No S command in the block containing G96 or G97

Index	:	TURRET
Character-string	:	None
Code	:	1

2928 Data word: 'T' error

Illegal T command

In T******, respective two-digit numbers expressing tool number, tool offset number and tool nose radius compensation number are larger than 32.

Index : TURRET

Character-string: None

Code : Tool offset number (>32) in grooving cycle
Hexadecimal number of the programmed T word when it is not:

$\emptyset < T < 99999999$

When tool offset number, tool number or tool nose radius compensation number is greater than 32, hexadecimal number of that number is in the right four digits.

2929 Data word: 'X' error

Illegal X command

Numerical value of X word is not:

$-99999.999 \leq X \leq 99999.999$

Or the X word programmed in incremental word is not $-99999.999 \leq X \leq 99999.999$ when converted into the absolute value.

Index	:	TURRET
Character-string	:	None
Code	:	Hexadecimal number of the programmed X word

293Ø Data word: 'X', 'Z' error

The first block of the G31, G32 and G33 mode (thread cutting fixed cycle) has only either of X and Z commands, or it has neither X nor Z command.

In the G3Ø gauging cycle mode, both X and Z commands are programmed.

Index	:	TURRET
Character-string	:	None
Code	:	1

2931 Data word: 'Z' error

Illegal Z command

Numerical value of Z word is not:

$-99999.999 \leq Z \leq 99999.999$

Or the Z word programmed in incremental word is not $-99999.999 \leq Z \leq 99999.999$ when converted into the absolute value.

Index	:	TURRET
Character-string	:	None
Code	:	Hexadecimal number of the programmed Z word 1 .... Z-axis command is programmed in the G1Ø2 or G1Ø3 mode.

2932 Data word: arc cal. error

In direct arc radius command, the coordinates of the arc center can not be calculated from the L command and X and Z commands.

The command error between I and K commands and X and Z commands in circular interpolation exceeds the tolerance (std.: 2Ø μm).

Index : TURRET
Character-string: None
Code :
1 .... L value is too small and there is no circle passing the target point.
2 .... Overflow in calculation of arc center or error
3 .... Error between I and K commands and X and Z commands exceeds the tolerance.
4 .... Arc radius I is zero.
1Ø .... End point of the arc command after calculation of LAP, tool nose radius compensation, or tool offset is offset from the programmed arc more than the specified tolerance.

2933 Data word: angle error

In the GØØ, GØ1, GØ2, GØ3, G34 or G35 sequence, an A command is programmed both with X and Z commands.

The target point calculated from the angle does not fall within a range of -99999.999 and 99999.999.

In the G31 or G33 sequence, both A and I commands are programmed.

In the G32 sequence, both A and K commands are programmed.

The target point in the thread cutting fixed cycle calculated from the angle command does not fall within a range of -99999.999 and 99999.999.

Index : TURRET
Character-string: None
Code :
1 .... Both X and Z commands are programmed, or I or K command is programmed.
2 .... Neither X nor Z command is programmed.
3 .... The target point calculated from the angle command does not fall within a range of -99999.999 and 99999.999.
Other ... Hexadecimal number of the target point calculated from angle command A in thread cutting fixed cycle

2934 Data word: radius error

Either I and/or K command is programmed, or no X and Z commands are programmed with L command.

No L command in the G1Ø2 and G1Ø3 sequence

Index : TURRET
Character-string: None
Code : 1 .... I or K command is programmed.
2 .... Either X or Z command is not programmed, or neither X nor Z command is programmed.
3 .... No L command in the G1Ø2 and G1Ø3 sequence

2935 Data word: C command error

Alarm of C command

Numerical value of programmed C command calling for spindle orientation is either $C < Ø°$, or $C \geqq 36Ø°$.

A C command is programmed at B-turret side on a multi-machining model.

A C command is programmed in other than GØØ, GØ1, G5Ø, G1Ø1, G1Ø2 and G1Ø3 modes.

A C command value is outside of $-36Ø° < C < 36Ø°$.

Index : TURRET
Character-string: None
Code : 1 .... C command at B-turret side
2 .... C command in other than GØØ, GØ1, G5Ø, G1Ø1, G1Ø2 and G1Ø3
3 .... C command calling for zero movement in the G1Ø1 mode
Other ... Hexadecimal number of programmed C value

2936 Multi cycle: B illegal order error

$B < Ø°$ or $B \geqq 18Ø°$

In G71, G72 thread cutting cycle, tangent (B/2) is negative or resulted in overflow.

Index : TURRET
Character-string: None
Code : Hexadecimal number of mantissa of floating-point of tangent (B/2)

2937 Multi cycle: D illegal order error

In G71, G72, G73 or G74 mode, no D command is programmed or numerical value of D command is not:

$\emptyset < D \leq 99999.99$

Index :	TURRET	
Character-string:	None	
Code :	FFFFFFFF	No D command
	Other	Hexadecimal number of D value

2938 Multi cycle: F illegal order error

In G71, G72, G73 or G74 mode, no F command is programmed or numerical value of F command is not:

$\emptyset < F \leq 99999.99$

Index :	TURRET	
Character-string:	None	
Code :	FFFFFFFF	No F command
	Other	Hexadecimal number of F value

2939 Multi cycle: H illegal order error

In G71 and G72 thread cutting mode, no H command is programmed or numerical value of H command is not:

$\emptyset < H \leq 99999.99$

Index :	TURRET	
Character-string:	None	
Code :	FFFFFFFF	No H command
	Other	Hexadecimal number of H value

294Ø Multi cycle: H-U (W) less than D (M73) error

In M73 of G71 or G72 thread cutting mode, the value "H-U (W)" is smaller than D.

Index :	TURRET
Character-string:	None
Code :	None

2941 Multi cycle: parameter I, K over error

In the G73 and G74 grooving cycle, the parameter of I or K is greater than the allowable value causing negative groove depth.

Index : TURRET
Character-string: None
Code : None

2942 Multi cycle: I, K illegal order error

In the G71 mode, K command is programmed, both A and I commands are programmed, or neither A nor I command is programmed.

In the G72 mode, I command is programmed, both A and K commands are programmed, or neither A nor K command is programmed.

In G73 and G74 grooving cycle, I and K are not:

$\emptyset \leq I, K \leq 99999.99$

Index : TURRET
Character-string: None
Code : None .... Either K command is programmed in G71 mode, or I command in G72 mode.
1 .... Either both A and I commands are programmed, or neither A nor I command is programmed in G71 mode, or either both A and K commands are programmed, or neither A nor K command is programmed in G72 mode.
Other ... Hexadecimal number of I or K value

2943 Multi cycle: L illegal order error

In G73 and G74 grooving cycle, numerical value of L command is not:

$\emptyset < L \leq 99999.99$

Index : TURRET
Character-string: None
Code : Hexadecimal number of L value

2944 Multi cycle: entry in LAP error

During LAP control, multi cycle command is programmed.

Index : TURRET
Character-string: None
Code : None

2945 Multi cycle: U (W) illegal order error

In G71 thread cutting cycle, either W command is programmed or the numerical value of U command is not:

$Ø < U \leq 99999.99$

In G72 thread cutting cycle, either U command is programmed or the numerical value of W command is not:

$Ø \leq W \leq 99999.99$

Index	:	TURRET
Character-string:		None
Code	:	None .... In G71, W command is programmed, or in G72, U command is programmed.
		Other ... Hexadecimal number of U (W) value

2946 Multi cycle: U (W) greater than H error

In G71 or G72 thread cutting cycle, programmed finish allowance U or W is larger than the thread height H.

Index	:	TURRET
Character-string:		None
Code	:	None

2947 Multi cycle: X, Z illegal order error

In G71, G72, G73 or G74 mode, either X or Z command is not programmed, or the value of them is not:

$-99999.999 \leq X\ (Z) \leq 99999.999$

Index	:	TURRET
Character-string:		None
Code	:	FFFFFFFF .... Either X or Z command is not programmed.
		Other .... Hexadecimal number of X or Z value

2948 Multi cycle: angle error

In the G71, G72 thread cutting cycle, A command is illegal and floating point of thread radius difference cannot be calculated.

Index	:	TURRET
Character-string	:	None
Code	:	Bit Ø ... Overflow in converting into integer Bit 1 ... Exponential underflow Bit 2 ... Exponential overflow Bit 3 ... Calculation of root of a negative number Bit 4 ... Division by Ø Bit 5 ... Angle overflow for TAN

2949 Multi cycle: tool offset error

In the G73, G74 grooving cycle, tool offset value specified in the program differs form the programmed shift direction.

Index	:	TURRET
Character-string	:	None
Code	:	Hexadecimal number of tool offset shift amount

295Ø Multi cycle: cycle start point error

In the G71, G72 thread cutting cycle, H command is too large and the reference point of thread cutting is not located in the infeeding direction from the cycle start point.

Index	:	TURRET
Character-string	:	None
Code	:	None

2951 Multi cycle: entry in NOSE-R error

During tool nose radius compensation mode, multi cycle (compound fixed cycle) is programmed.

Index	:	TURRET
Character-string	:	None
Code	:	None

2952 Multi cycle: width error

In the G73, G74 grooving cycle, the tool width calculated from the tool offset value is larger than the groove width.

Index	:	TURRET
Character-string	:	None
Code	:	Hexadecimal number of final grooving amount

2953 Chamfering: GØ1 mode error

Chamfering commands are programmed in other than GØ1 mode.

Index	:	TURRET
Character-string	:	None
Code	:	None .... No G code programmed
		2 .... GØ2
		3 .... GØ3
		1F G31
		2Ø G32
		21 ...... G33
		22 ...... G34
		23 ...... G35
		FE ...... GØØ

2954 Chamfering: parameter L over error

In chamfering commands, programmed L value is larger than the axis movement distance.

Index	:	TURRET
Character-string	:	None
Code	:	Hexadecimal number of axis movement distance

2955 Chamfering: L illegal order error

In chamfering commands, no L command is programmed, or programmed L value is not:

$-99999.999 \leq L \leq 99999.999$

Index	:	TURRET
Character-string	:	None
Code	:	Hexadecimal number of L value

2956 Chamfering: X, Z illegal order error

In chamfering commands, either both X and Z commands are programmed, or neither X nor Z command is programmed.

Programmed X or Z value is not:

$-99999.999 \leq X\ (Z) \leq 99999.999$

Index	:	TURRET
Character-string	:	None
Code	:	FFFFFFFF ..... Either both X or Z command is programmed, or neither X nor Z is programmed.
		Other .... Hexadecimal number of X or Z value

2957 LAP: B illegal order error

B command specifying the tool tip angle in G88 LAP mode is either B < Ø° or B $\geq$ 18Ø°.

Index	:	TURRET
Character-string	:	None
Code	:	None

2958 LAP: D illegal order error

In G85, G86 and G88 LAP mode, either no D command is programmed, or the programmed D value is either negative or there are too many digits.

Index	:	TURRET
Character-string	:	None
Code	:	1 .... Programmed D value is not: $-99999.999 \leq D \leq 99999.999$
		2 .... Either negative or zero
		3 .... No D command programmed

2959 LAP: DA (DB) illegal order error

In G85 LAP mode, when XA (ZA) or XB (ZB) is programmed with G84, programmed value of either DA or DB is negative or there are too many digits.

Index	:	TURRET
Character-string	:	None
Code	:	1 .... Programmed value of DA of DB is not: $-99999.999 \leq DA\ (DB) \leq 99999.999$
		2 .... Either negative or zero

2960 LAP: H illegal order error

In G88 LAP mode, no H command is programmed, or programmed H value is negative or there are too many digits.

Index	:	TURRET
Character-string	:	None
Code	:	1 .... Programmed H value is not: $-99999.999 \leqq H \leqq 99999.999$
		2 .... Either negative or zero
		3 .... No H command programmed

2961 LAP: H-U (W) less than D (M73) error

In M73 of G88 LAP mode, the value "H-U (W)" is smaller than D, and finish cut cycle is impossible.

Index	:	TURRET
Character-string	:	None
Code	:	None

2962 LAP: U (W) illegal order error

The U or W command value in G85, G86 G87, G88 LAP mode is negative or larger than the allowable maximum value.

Numerical value of U and W commands must be:

$0 \leqq U\ (W) \leqq 99999.999$

Index	:	TURRET
Character-string	:	None
Code	:	1 .... Programmed U or W value is not; $-99999.999 \leqq U\ (W) \leqq 99999.999$
		2 .... Programmed U or W command value is negative.
		3 .... Infeeding direction is reversed by finish allowance.

2963 LAP: U (W) greater than H error

In G88 LAP mode, programmed finish allowance U or W is larger than the thread height H.

Index	:	TURRET
Character-string	:	None
Code	:	None

2964 LAP: XA (ZA), XB (ZB) illegal order error

In G85 LAP mode, when G84 is programmed, the number of digits of the programmed XA (ZA) or XB (ZB) command is larger than the allowable number.

Index : TURRET
Character-string: None
Code : 1 .... In longitudinal cycle, programmed XA or XB command is not:
$-99999.999 \leq XA (XB) \leq 99999.999$
2 .... In transverse cycle, programmed ZA or ZB command is not:
$-99999.999 \leq ZA (ZB) \leq 99999.999$

2965 LAP: calculation error

Calculation alarm during LAP processing (when calculating arc center and radius in G85).

Index : TURRET
Character-string: None
Code : XX:

Bit Ø ... Overflow in converting into integer
Bit 1 ... Exponential underflow
Bit 2 ... Exponential overflow
Bit 3 ... Calculation of root of a negative number
Bit 4 ... Division by Ø
Bit 5 ... Angle overflow for SIN, COS, TAN and COT

2966 LAP: a number of down stair over error

In G85 LAP mode, the number of descending steps exceeds ten.

Index : TURRET
Character-string: None
Code : None

2967 LAP: entry in LAP error

During LAP control, LAP command (G85, G86, G87 and G88) is programmed.

Index : TURRET
Character-string: None
Code : None

2968 LAP: sequence name error

In the block containing G85 or G86 calling for LAP mode, no sequence name is programmed, or the programmed sequence name is not found in the program.

Index	:	TURRET
Character-string	:	None
Code	:	1 .... No sequence name is programmed in G85 or G86 block. 2 .... The block assigned with the sequence name specified is not found.

2969 LAP: LAP control error

LAP control is impossible.
(Overflow of control counter for LAP control)

Index	:	TURRET
Character-string	:	None
Code	:	None

297Ø LAP: G-code error

No G8Ø command up to the end of the program after G81 or G82 is programmed.

G81 or G82 is not programmed in the sequence assigned with the sequence name designated in the sequence containing G85 or G86.

Index	:	TURRET
Character-string	:	None
Code	:	1 .... No G8Ø programmed. 2 .... G81 or G82 is not programmed in the sequence assigned with the sequence name designated in the sequence containing G85 or G86.

2971 LAP: entry in NOSE-R error

While nose radius compensation mode is active, G code calling for LAP mode (G85, G86, G87 and G88) is programmed.

Index	:	TURRET
Character-string	:	None
Code	:	None

2972 NOSE-R comp.: calculation error

Error in floating-point calculation for nose radius compensation

Index : TURRET
Character-string: None
Code : XYY

YY:

Bit Ø ... Overflow in converting into integer
Bit 1 ... Exponential underflow
Bit 2 ... Exponential overflow
Bit 3 ... Calculation of root of a negative number
Bit 4 ... Division by Ø
Bit 5 ... Angle overflow for SIN, COS, TAN and COT

X:

1 .... Calculation of graphic factor of straight line
2 .... Calculation of graphic factor of arc
3 .... Offset calculation of graphic factor of nose radius compensation amount
4 .... Vertical vector calculation of straight lines and arcs
5 ....
6 .... Calculation of point of intersection: straight line and straight line
7 .... Calculation of point of intersection: straight line and arc
8 .... Calculation of point of intersection: arc and arc
9 .... Calculation to select the target point from possible two points of intersection with an arc
A .... Recalculation of graphic factor of arc
B .... Calculation of I and K from nose radius compensation point
C .... Calculation of commands X, Z, I and K

2973 NOSE-R comp.: cancel impossible error

Code G4Ø cancelling tool nose radius compensation mode is programmed in other than the GØØ or GØ1 mode.

Index	:	TURRET
Character-string	:	None
Code	:	None

2974 NOSE-R comp.: no cross point error

Point of intersection cannot be calculated in line to arc or arc to arc intersection.

Index	:	TURRET
Character-string	:	None
Code	:	1 .... Straight line to arc
		2 .... Arc to straight line
		3 .... Arc to arc

2975 NOSE-R comp.: no spec. error

G41 and G42 calling for tool nose radius compensation mode is programmed although the control has no nose radius compensation specification.

Index	:	TURRET
Character-string	:	None
Code	:	None

2976 NOSE-R comp.: start up impossible error

Tool nose radius compensation mode entry is intended in other than permissible manner, and compensated point cannot be calculated.

Index	:	TURRET
Character-string	:	None
Code	:	None .... G41 or G42 is programmed in other than GØØ or GØ1 mode.
		1Ø ...... The same X and Z commands are specified in the G41/G42 sequence following it.
		11 ...... X and Z command are not programmed in the block following G41/G42 block.
		4Ø ...... G4Ø is programmed in the block following G41/G42 block.

2977 NOSE-R comp.: thread cycle error

G31, G32 or G33 calling for thread cutting cycle is programmed during the tool nose radius compensation mode.

Index	:	TURRET
Character-string	:	None
Code	:	1F ...... G31 was programmed. 2Ø ...... G32 was programmed. 21 ...... G33 was programmed.

2978 Tool life control: no spec. error

Tool life management variables are programmed although the control has no tool life management specification.

Tool group TG and tool offset group OG are programmed.

Mnemonic G code or TLID is programmed.

Index	:	TURRET
Character-string	:	None
Code	:	1 .... Tool life management variables are programmed in the left part of the expression. 2 .... Tool life management variables are programmed in the right part of the expression. 5 .... Tool group TG and tool offset group OG are programmed. Mnemonic G code or TLID is programmed.

2979 Tool life control: tool group error

Numerical value of tool group command TG is: TG < 1 or TG > 13

Index	:	TURRET
Character-string	:	None
Code	:	Hexadecimal number of the programmed TG

298Ø Tool life control: no T-entry error

Tools are not registered in the programmed tool group.

Index	:	TURRET
Character-string	:	None
Code	:	FFFFFFFF

2981 Tool life control: tool offset group error

Illegal tool offset group number is programmed.

Index :	TURRET
Character-string:	None
Code :	Hexadecimal number of the programmed tool offset group number

2982 Tool life control: no T-offset error

Tool offset number is not registered for the programmed tool offset group.

Index :	TURRET
Character-string:	None
Code :	FFFFFFFF

2983 Cannot IGF-convert G82 (N****) error

2984 Cannot IGF-convert G32 (N****) error

2985 Cannot IGF-convert G72 (N****) error

The G72 code cannot be converted for OSP3ØØØ programs.

2986 Cannot IGF-convert B command (N****) error

If B (turret) is not "Ø", the G71 (longitudinal thread cutting compound cycle) cannot be converted for OSP3ØØØ program.

2987 Cannot IGF-convert L command (N****) error

If M23 (chamfering ON) and L (Chamfering amount) are used in the G71 (longitudinal thread cutting compound cycle), such a cycle cannot be converted for OSP3ØØØ program.

2988 Cannot IGF-convert K command (N****) error

299Ø Cannot IGF-convert TG command (N****) error

The TG (tool group) command cannot be converted for OSP3ØØØ program.

2991 Cannot IGF-convert OG command (N****) error

The OG (offset group) command cannot be converted for OSP3ØØØ program.

2993 LAP: NOSE-R not cancelled error

Nose radius compensation mode is not cancelled at the end of LAP (sequence containing G8Ø).

Index	:	TURRET
Character-string	:	None
Code	:	None

2994 NOSE-R comp.: NOSE-R > circle-R error

Point of intersection cannot be calculated since the radius of the programmed arc is smaller than nose radius.

Index	:	TURRET
Character-string	:	None
Code	:	1 .... The arc radius is smaller than the nose radius when obtaining the point of intersection - straight line to arc.
		2 .... The arc radius is smaller than the nose radius when obtaining the point of intersection - arc to straight line.
		3 .... The arc radius is smaller than the nose radius when obtaining the point of intersection - arc to arc.
		4 .... The arc radius in the sequence following the G41/G42 sequence is smaller than nose radius.

2999 Output buffer over error

During the conversion, output program buffer becomes full.

3ØØ5 Data word: SB error

Programmed SB command is not:

$\emptyset \leqq SB \leqq 9999$

SB command is programmed for B-turret.

Index	:	TURRET
Character-string	:	None
Code	:	1 .... SB command is programmed for B-turret.
		Others .. Hexadecimal number of commanded SB value

3ØØ6 Fixed cycle: no spec. error

G code calling fixed cycle for multi-machining model is programmed for a lathe without multi-machining function.

Index	:	TURRET
Character-string:		None
Code	:	1

3ØØ7 Fixed cycle: C error

Programmed C value is not:

$-36Ø° < C < 36Ø°$

Index	:	TURRET
Character-string:		None
Code	:	Hexadecimal number of programmed C value

3ØØ8 Fixed cycle: I, K error

In G181 through G184 and G189 mode cycle, both I and K or neither I nor K is programmed.

In G181 through G184 and G189 mode cycle, programmed I and K values are not:

$Ø \leqq I, K \leqq 99999.999$

In G185 through G188 mode cycle, programmed I and K values are not:

$-99999.999 \leqq I, K \leqq 99999.999$

Index	:	TURRET	
Character-string:		None	
Code	:	1	Both I and K commands are programmed.
		FFFFFFFF	No I and K commands
		Other	Hexadecimal number of I and K values

3ØØ9 Fixed cycle: Q error

Programmed Q value is not:

$1 \leqq Q \leqq 9999$

Index	:	TURRET
Character-string:		None
Code:	:	Hexadecimal number of Q value

3Ø1Ø Fixed cycle: F

Programmed F value is either Ø or negative.

No F command

Index	:	TURRET
Character-string	:	None
Code	:	FFFFFFFF ..... No F command Other .... Hexadecimal number of programmed F value

3Ø11 Fixed cycle: L

Programmed L value is not:

$Ø < L \leq 99999.999$

Index	:	TURRET
Character-string	:	None
Code	:	Hexadecimal number of programmed L value

3Ø12 Fixed cycle: D

Programmed D value is not:

$Ø < D \leq 99999.999$

Index	:	TURRET
Character-string	:	None
Code	:	FFFFFFFF ..... No D command Other .... Hexadecimal number of programmed D value

3Ø13 Fixed cycle: X, Z

In the block containing G181 through G189, either X or Z is not programmed. Or, numerical value of X and Z is not:

$-99999.999 \leq X, Z \leq 99999.999$

Index	:	TURRET
Character-string	:	None
Code	:	FFFFFFFF ..... No X and Z command Other .... Hexadecimal number of programmed X and Z value

3Ø14 Fixed cycle: SA error

Programmed SA value is not:

Ø < SA ≦ 2Ø

No SA command

Index :	TURRET	
Character-string:	None	
Code :	FFFFFFFF	No SA command
	Other	Hexadecimal number of programmed SA value

3Ø15 Fixed cycle: feed G94 error

G185 through G188 is programmed in the G94 mode.

Index :	TURRET
Character-string:	None
Code :	None

3Ø17 Fixed cycle: thread cycle error

In thread cutting cycle, programmed I or K value is too large.

Index :	TURRET
Character-string:	None
Code :	1

3Ø18 Data word: QA error

QA command is programmed for B-turret.

QA command is programmed in other than the GØØ and GØ1 mode.

Programmed QA value is not:

Ø ≦ QA ≦ 1999

Index :	TURRET	
Character-string:	None	
Code :	1	QA command is programmed for B-turret.
	2	QA command is programmed in other than GØØ and GØ1 mode.
	Other ...	Hexadecimal number of QA value

3Ø19 Data word: X, Y command error

In coordinate system conversion, both X and Y are assigned "zero".

Index	:	TURRET
Character-string	:	None
Code	:	None

3Ø2Ø Data word: incremental error

In the coordinate system conversion, the G code (G91) calling incremental mode is programmed.

Index	:	TURRET
Character-string	:	None
Code	:	Hexadecimal number of commanded code

3Ø21 Data word: no X, Y error

In coordinate system conversion, only one of X and Y is programmed.

Index	:	TURRET
Character-string	:	None
Code	:	None

3Ø22 Data word: 'Y' error

In coordinate system conversion, the numerical value of "Y" is not within the range below:

$$-99999.999 \leqq Y \leqq 99999.999$$

Index	:	TURRET
Character-string	:	None
Code	:	Hexadecimal number of commanded Y value

3Ø23 Data word: distance cal. error

In coordinate system conversion, the value after the conversion is larger than 99999.999.

Index	:	TURRET
Character-string	:	None
Code	:	None

3Ø24 Data word: 'R' error

In a block containing G181 through G184 or G189, either RØ command is programmed or an R command is programmed with X and Z commands.

Or the numerical value of X, Z or R does not fall within the following range:

$-99999.999 \leqq X, Z, \text{ or } R \leqq 99999.999$

Index :	TURRET	
Character-string:	None	
Code :	1........	Simultaneous programming of R with X and/or Z
	FFFFFF ..	No X, Z or R command
	Others ..	Hexadecimal number of numerical value of programmed X, Z or R

Note 1: For messages not listed above, refer to the messages explained in Appendix 1, Alarm B of Maintenance Manual for OSP5ØØØ.

Note 2: Message "Cannot IGF-convert" might be displayed if a program file not created using IGF is designated as the input file.